AN INTELLIGENT PLANNING TOOL FOR EVALUATION OF AMPHIBIOUS RIVER CROSSING SITES

by

Mark W. Yenter

B.S., University of Nevada, 1981

A thesis submitted to the

Faculty of the Graduate School of the

University of Colorado in partial fulfillment

of the requirements for the degree of

Master of Science

Department of Civil, Environmental,

and Architectural Engineering

1990

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave Blank)*

2. REPORT DATE
9 Feb 90

3. REPORT TYPE AND DATES COVERED
Final

4. TITLE AND SUBTITLE
An Intelligent Planning Tool for Evaluation of Amphibious River Crossing Sites

5. FUNDING NUMBERS

6. AUTHOR(S)
CPT Mark William Yenter

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
HQDA, MILPERCEN (DAPC-OPB-D)

8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unlimited

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

The use of two dimensional hydrodynamic finite element models to evaluate potential amphibious river crossing sites.

14. SUBJECT TERMS
two dimensional hydrodynamic modeling, river crossing operations, finite element networks, Bradley Fighting Vehicle.

15. NUMBER OF PAGES

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT
UNCLASSIFIED

18. SECURITY CLASSIFICATION OF THIS PAGE
UNCLASSIFIED

19. SECURITY CLASSIFICATION OF ABSTRACT
UNCLASSIFIED

20. LIMITATION OF ABSTRACT
SAR

This thesis for the Master of Science degree by

Mark W. Yenter

has been approved for the

Department of

Civil, Environmental, and Architectural Engineering

by

Kenneth M. Strzepek

Steven C. Chapra

Date 2/9/90

Yenter, Mark William (M.S. Civil Engineering)
An Intelligent Planning Tool for Evaluation of Amphibious River Crossing Sites
Thesis directed by Research Associate Professor Kenneth M. Strzepek

In the past, two-dimensional hydrodynamic finite element models have been used to successfully model water current velocity and surface elevation. Because they are mathematical models, they have are a static, that is, initial boundary conditions are set and the model is run. The question is, can the results of static mathematical hydrodynamic modeling be extended to dynamic simulation?

In this thesis a decision support system is developed that applies the static results of two-dimensional hydrodynamic modeling in a graphics oriented, dynamic computer simulation of an amphibious vehicle crossing a river. As a decision support system, the program permits the user to build the simulation scenario by selecting the crossing site, vehicle type that will attempt the crossing, water current, and starting and finish points. The decision support system, called the River Crossing Site Simulation and Evaluation Tool (RC-SET), assists the

user in evaluating the feasibility of crossing a river under specific hydrologic conditions with a known vehicle type.

This thesis is dedicated to

Lisa,

my best friend.

ACKNOWLEDGEMENTS

Many people have assisted me in my efforts. My thanks to Dr. Kenneth M. Strzepek who helped me to develop and focus this thesis, and whose imagination and sense of humor have been an inspiration. My thanks also go to the other members of my graduate committee, Dr. Steve Chapra, Dr. Earnest Flack, and Dr. Tissa Illangasekare, all are academic professionals of the highest caliber.

Many thanks to my friends and colleagues at the Center for Advanced Decision Support for Water and Environmental Systems. Tom Over and Edith Zagona where always there when I needed advice about water resources issues or a shot in the arm of inspiration. Terri Betancourt introduced me to Geographical Information Systems and was a dependable and never ending source of common sense and good ideas. My deepest thanks to David Sieh, Brian Makare, and Steve Wehrend whose interesting combination of saintly patience and man-handling helped to transform me from a broken FORTRAN IV programmer into a respectable FORTRAN 77 hacker. My apologies to Jim Waterman, the computer systems guy,

whose disk space I mercilessly poached during the last year.

A few people outside of the University of Colorado provided me with information and software, without which this thesis would not be possible. Thanks to Dr. D. Michael Gee from the Corps of Engineers Hydrologic Engineering Center for providing the source code and documentation of the two dimensional model, and the final report of the Sacramento River study. My thanks to Captain Bo Underwood from the U.S. Corps of Engineers Waterways Experiment Station for taking the time to locate and send to me the field testing data on a number of tracked vehicles.

And finally, thanks to my wife Lisa, and our darling daughter Lindsay Marie, for putting up with my absence, both mental and physical, during the last eighteen months.

CONTENTS

TABLES

Table

FIGURES

Figure

CHAPTER I

INTRODUCTION

The River Crossing Problem

Current doctrine. During the last decade the United States Army has recognized that the size of enemy formations and the lethalness of modern weapons dictated a change in the way we would fight the next war. Our army is much smaller than our major opponents, but is better trained and equipped. The new army doctrine seeks to overcome this disadvantage in numbers with aggressive operations that will allow us to maintain the initiative on the battlefield in order to dictate how the battle is to be fought. This new doctrine is known as the Airland Battle Doctrine.

The Airland Battle Doctrine identifies three areas of combat where the battle will be waged. Close Operations take place at the leading edge of the friendly forces, where they engage and destroy the leading elements of the enemy. Deep Operations are directed at the follow-on forces of the enemy and are conducted to destroy and disrupt reinforcing units to prevent them from supporting forward units. Rear

prevent them from supporting forward units. Rear Operations are those activities conducted in the rear area of friendly forces and are designed to ensure continuity of friendly operations (U.S.Army, 1989).

River crossing operations. The ability of the U.S. Army to cross rivers quickly and efficiently while conducting Close Operations is essential to the success of the Airland Battle Doctrine (U.S.Army, 1984). Despite advances in high mobility weapon systems and extensive aviation assets, rivers remain major obstacles. River crossings are among the most critical, complex, and vulnerable combined arms operations (U.S.Army, 1984).

There are three types of river crossing operations; hasty, deliberate, and retrograde. A hasty river crossing is a decentralized operation that is conducted as a continuation of an attack by friendly forces. It is used to cross rivers where the enemy forces are weak or disrupted, and in situations where the characteristics of the river are such that the stream is not a major obstacle. The methods used to conduct hasty river crossings include the use of existing civilian bridges and ferries, ford sites, amphibious vehicles, and military rafting and bridging equipment.

Deliberate river crossing operations are conducted in situations where hasty crossing are not practical or have failed. Deliberate river crossing operations require detailed planning and the buildup of assaulting forces, river crossing assets, and fire support. It is a three-phased operation consisting of the assault phase, the rafting phase, and the bridging phase. The selection of crossing, rafting, and bridging sites are critical to the success of the operation (U.S.Army, 1984).

In the assault river crossing phase the commander of the friendly forces attempts to cross sufficient forces to secure the far bank of the river. Assault river crossing sites must be located where enemy forces are weak, there is concealment from enemy observation, and friendly forces occupy the dominant terrain features. This insures security and supporting fire for the assaulting forces. Adequate routes to the river, as well as from the river to follow-on objectives are also required. The river current velocity must be small, less than five feet per second, and the river should be crossed at a narrow point to minimize the exposure of friendly forces to hostile fire.

When amphibious vehicles are used the ingress and egress bank conditions are very important. As a

general rule bank slopes should have a slope of 30 percent or less. Ideally the banks soil is firm and capable of handling high traffic, allowing multiple passes of ingressing and egressing vehicles.

If fording sites are available they must be shallow with a water depth not greater than 39 inches for dismounted soldiers and light armored vehicles, and not greater than 42 inches for medium to heavy armored vehicles. The ingress and egress banks must have slopes of 30 percent or less if armored vehicles are used. The riverbed must be firm enough to withstand the vehicle traffic.

During the rafting phase friendly forces reinforce the assaulting forces by rafting armored vehicles and anti-armor weapons. Rafting sites are located where they can provide the fastest access to the far shore. Rafting sites require sufficient road networks leading up to them on the near shore and adequate exit routes on the far shore to permit hasty movement. Both banks should be firm and have slopes not greater than 20 percent. If possible the rafting site should be located at a narrow part of the river, but must be free of sandbars or other obstacles the would impede rafting operations. The draft requirements of the four types of rafting equipment currently in the Army inventory vary from 22 inches to

29 inches. Rafting operations can be conducted in water current velocities as high as 10 feet per second, but the practical limit is less than or equal to 5 feet per second. Because there is a risk of rafts being swept downstream, rafting sites are always located downstream of bridging sites.

During the bridging phase the bulk of the advancing friendly forces cross the river. Organic (those normally found in the unit) and attached bridging assets are used to construct temporary floating bridges that allow wheeled and remaining tracked vehicles to cross the river. Bridging sites must be located where adequate road networks already exist or can be rapidly constructed. The bank slopes, either existing or constructed, cannot be greater than 10 percent. The bank soil must be compacted and capable of handling high traffic, either in it natural state or stabilized through standard construction practices or expedient methods such as temporary matting. The velocity of the water cannot exceed 10 feet per second, and the minimum draft requirements for military float bridging equipment range from 24inches to 40inches. Floating bridges also require assembly sites along the water's edge where pontoons and other components can be pre-assembled then maneuvered into position for anchoring. Obviously, the bridging site

must be located at the narrowest point along the river that meets the above criteria.

Retrograde river crossing operations are different from hasty and deliberate crossings because they are defensive in nature. They are conducted when enemy forces threaten to overwhelm friendly forces. They are carefully planned operations that are designed to trade space for time. Retrograde river crossing operations are planned to successfully extract friendly forces from the enemy shore and force the enemy to conduct a deliberate crossing.

River crossing engineer information requirements. On the battlefield the area commander determines the specific area of operations along which he wants to conduct the river crossing operation. His decision is based upon tactical considerations such as enemy strength, key terrain, and cover and concealment of friendly forces. The burden of determining the validity and practicality of the crossing sites selected falls upon the Corps of Engineers' officers supporting the maneuver commander. As discussed above, several characteristics of potential river crossing sites should be known to evaluate their feasibility from an engineering standpoint. Table 1.1 lists the

type of information that is required to evaluate a selected river crossing site.

TABLE 1.1 River Crossing Information Requirements

Crossing Method	HYDROLOGIC		GEOTECHNICAL		
	Water Velocity	Water Depth	Bank Slopes	Bank Soil Conditions	Riverbed Soil Conditions
Fording	X	X	X	X	X
Amphibious Vehicles	X		X	X	
Rafting	X	X	X	X	
Float Bridging	X	X	X	X	

Table 1.1 dipicts the two general categories of information are required, hydrologic measurements of the water velocity and depth, and geotechnical qualification of the bank and riverbed soil conditions. Geotechnical issues are important, but beyond the scope of this thesis. Army engineers have a variety of sources of information to evaluate the hydrologic characteristics of the river.

Historical river data is the most dependable source of hydrologic information. However, this information may be limited or not available. When it is not available, or to update and augment this information, engineer units will typically send engineer soldiers to conduct a river reconnaissance. They will probe forward and gather hydrologic and geotechnical information. To maintain cover and concealment from enemy fire, their reconnaissance is limited to the bare essentials. River width is usually approximated using a compass to determine the azimuth to a point on the far shore, adding 45 degrees to this azimuth and siting the same point again on the new azimuth from a second position. The distance between the two positions on the near bank is the approximate width of the river. When practical, river depth is measured using expedient rods or ropes tied to weights.

Water velocity is approximated by timing a floating object, such as a piece of wood, as it travels over a measured distance (U.S.Army, 1985). The hydrologic data gathered by these soldiers is sketchy at best, but is extremely important when no historical records are available.

Topographic engineer units may provide even less specific information about the river. On larger rivers it is sometimes possible to make a rough estimates of the water velocity from aerial photographs (U.S.Army, 1978). This is done by taking consecutive photographs of an object floating in the river, such as a log, and noting the lapse time between them. The appearance of the river can be an important indicator of river velocity. Meandering rivers generally have average water velocities of 4.5 feet per second or less.

In some cases, particularly where the river was originally in friendly territory before the conflict, historical records of river stage and corresponding flow rates may be available in the form of rating curves or tabular data (Chow, 1988). Water velocity data would be less likely to be available but would be useful to evaluate specific locations, and could be used to approximate velocities at other sites in the

river if the reach geometry changes very little, and the length of reach is fairly long.

Even though specific velocity information may be available, rarely will it be in the quantity or detail needed to indicate the velocity distribution throughout the reach. Complex flow patterns can exist where the channel flow is split by sandbars and islands, flows over and through submerged obstacles such as destroyed bridges and rock formations, and flows around sharp turns in the river. Engineer officers are required to make a best estimate of the river velocity distribution at different locations throughout the river reach, often from very little data.

Needed is a method for determining the two dimensional flow patterns of the river at the existing flow rate and reach topography. This may not be possible for those rivers located in enemy territory before the outbreak of hostilities, however, flow rate and corresponding velocity distribution data could be compiled for rivers in friendly territory. These values could be approximated using any of a number of existing two dimensional hydrodynamic models.

With this information, engineer officers could evaluate potential crossing sites with greater accuracy. The magnitude and direction of the velocity vectors could be compared to the maximum values discussed above to identify amphibious crossing, fording, rafting, and bridging points. It would be even more beneficial to apply the two dimensional velocity data in a computer simulation of the force of the river acting against vehicles and equipment as they perform their river crossing tasks in order to test the feasibility of a selected sites. A simulation of this sort would certainly reduce the risk of these precarious operations.

Focus of This Thesis

The purpose of this thesis is to determine the feasibility of using a two dimensional finite element hydrodynamic model as the basis of a decision support tool that can be used to evaluate potential amphibious assaulting sites.

The thesis will address hydrologic aspects of the amphibious river crossing problem. The geotechnical qualification of the bank slopes and trafficability are important considerations, but their study is beyond the scope of this thesis.

This thesis is organized into six chapters. The river crossing problem and current U.S. Army doctrine has been addressed in Chapter I. The type of engineering information required to evaluate potential amphibious river crossing sites was also identified. Chapter II is a review of current literature about decision support systems, simulation, two dimensional finite element models in general, and more specifically, finite element modeling system operation. The hydrodynamic model RMA-2D is presented in Chapter III. In Chapter IV the organization and methodology of the prototype decision support system used to simulate and evaluate potential amphibious river crossing sites is discussed in detail. The prototype system is called the River Crossing Site Simulation and Evaluation Tool (RC-SET). The results of applying RC-SET in a case study are presented in Chapter V. In this case study RC-SET is used to simulate an amphibious vehicle crossing a river reach. Chapter VI summarizes the development and application of the prototype decision support system. Continuing research and development of RC-SET are also addressed.

CHAPTER II

LITERATURE REVIEW

Decision Support Systems

Before evaluating the feasibility of developing a computer simulation of a potential river crossing site we need to define terms and discuss the concepts of decision support systems, modeling, and computer simulation. Today, computer programs are labeled as decision support systems without much thought as to whether or not they actually are such systems. By definition, a decision support system (DSS) is an interactive computer-based system designed to help decision makers use data and models to solve unstructured problems (Hopple, 1988).

Although DSSs vary greatly in fields ranging from corporate planning to microbiology, they share common characteristics. DSSs assist the user in making decisions, usually on semistructured problems. They are a tool used by humans to focus the decision making process, and do not replace human judgment. They process, organize, and interpret voluminous quantities of information that would otherwise overwhelm the user,

and by doing so enable the user to make decisions more effectively, and more efficiently. DSSs are designed to be used by people with little or no computer experience. They should make the user feel comfortable with the system and not intimidated by it (Hopple, 1988). They also provide continuous interactive problem solving rather than off-line batch processing. DSSs should not impose or force a decision making process on the user. Rather, the DSSs should provide the user with a selection of blocks to build the decision making process. Additionally, DSSs should be designed in a modular system so that they can be readily updated and improved as the designer's problem solving methods change. This allows the designer to change one portion of the DSS without redesigning the entire system.

DSSs consist of three subsystems: a user-system interface; a database management subsystem; and a model base management system (Hopple, 1988). The user-system interface allows the user to communicate with the components of the DSS. The interface can be one or more of a variety of input and output schemes generally categorized as the action language, the display or presentation language, and the knowledge base. The user uses the action language to communicate with the DSSs via keyboard, mouse, touch activated screen,

joystick or even voice command. DSSs use display or presentation language to communicate with the user via graphics on the screen, audio output, line printers, and plotters. The knowledge base is that information required by the user to use the DSS properly. If not already known by the user, this information may be available through user's manuals, or on-line help keys.

The database management subsystem is a computer program designed to facilitate the management of integrated collection of data (Hopple, 1988). Database subsystems are particularly important when the DSS will be used to evaluate unstructured or loosely structured problems where extensive and frequent data restructuring will be required. Database management programs must also have the capability to load and manage external files.

The third subsystem of a DSS is the model base management system. A model is a simplified representation of reality or some aspect of it. A model based management system is a computer program that offers a wide range of models and allows for flexible access, update, and change to the model base (Hopple, 1988). The model base management system provides the user with the building blocks that are used to construct variations of the model system.

<u>Simulation</u>

A decision support system is built around a model that simulates the system to be evaluated. In common language, to simulate something means to assume the appearance of that thing. However, a more categorical meaning is required in engineering and science. In these terms "a simulation is the forming of an abstract model from a real situation in order to understand the impact of modifications and the effect of introducing various strategies," (Negoita, 1987). The main objective in simulation is to aid the user in determining what will happen in a given physical situation under certain simplifying assumptions. The major advantage of simulations is that they provide the user with a means of experimenting without destroying, damaging or, in any way, modifying the real system.

Modeling. To simulate a series of actions, a model must be produced. Models have been defined in many ways by many scholars, but an adequate definition is that a model is a formal, symbolic representation of a system (Lewis and Smith, 1979). Models can be used to approximate reality because both the model and the real system follow the same physical laws, however, models are simplifications of reality. The more closely the model approximates reality, the more complex the model

becomes. "Models should never be confused with reality. They just map some facets of reality (hopefully the ones we had in mind) into an abstract description," (Cellier, 1982).

Before the advent of computers, physical models were probably the most commonly used models. Examples of physical models are wind tunnels used in aircraft and automobile design, and water tanks used in hydraulic studies and ship design. The benefits of physical models are that they are relatively easy to construct and can model complex physical phenomena, such as the formation of ice on submerged structures, that would otherwise be very difficult to model. Physical models do have disadvantages. One disadvantage is that they are highly specific. A physical model of one port facility will not be valid for another (Pidd, 1988). Another disadvantage is that experimentation with physical models usually requires alteration of the model, and therefore, expensive and frequent construction of variations of the model.

Scale can be a disadvantage in physical modeling. While it may be relatively easy to construct a scale replication of the real system, such as a highway bride, it may not be possible to accurately model very small systems. For example, consider a scale model of flow through a porous medium. In this

case, the scale model is an enlargement of the real system. While it may be possible to model the physical cavity space between typical grains of sand, but it will not be possible to model the capillary forces exerted by the water relative to the size of the cavity.

Since the mid-sixties computers have been used successfully to model mathematically engineering and applied science systems. Mathematical models are effective in classical engineering and physics where physical phenomena is understood and can be defined by equations and empirical coefficients. The initial success of these models led many scholars to believe that they could be applied to other sciences like biology, economy, sociology, and psychology (Cellier, 1982). Such was not the case. In the mid-seventies engineers studying water resources and environmental systems realized that they were difficult to describe in precise mathematical terms. To compensate for this "softness" or "grayness," fuzzy logic methods were developed. However, these fuzzy logic concepts prove to be inadequate for ill-defined systems (Cellier, 1982).

Computer simulation as experimentation. Computer simulation is experimentation using a computer-based

model of some real system. The model is used as the vehicle for experimentation, and is often a trial and error way to test the effects of various policies when applied to the real system (Pidd, 1988). Just as mathematical models cannot model all systems, computer simulations are not a panacea. Realistic simulations require long computer programs that can be quite complex. The more the model is made to approximate reality, the greater the overhead in time, effort, and hardware resources. However, in most cases computer simulations have distinct advantages over real experimentation and mathematical modeling (Pidd, 1988):

Cost. Computer simulations can be more cost effective than direct experimentation. Even though computer simulation can be time consuming and expensive in terms of skilled manpower, real experiments can also be time consuming and expensive, particularly if the real system must be modified.

Time. Computer simulations provide profitable return for the initial time investment of producing working computer programs for simulation models. The return on this initial investment is that computer simulations can simulate weeks, months or even years in a few seconds of computer time.

Replication. Unlike physical experiments, computer simulations are precisely repeatable. Physical experiments rarely yield the exact results when repeated due to errors and physical variations in the system. Computer simulations also allow the experiment to be conducted repetitively without changing the physical system.

Safety. Computer simulations allow us to test extreme conditions that would otherwise be hazardous.

Flexibility. Computer simulations are superior to mathematical models because they can be designed to cope with transient and dynamic effects.

The simulation process. The simulation process can be categorized into five phases: (1) systems analysis, (2) program synthesis, (3) model verification, (4) model validation, and (5) model analysis (Lewis and Smith, 1979). In practical terms, these phases are not distinct or separated from each other. The simulation process is iterative, starting with a general thesis and evolving into a final product as new requirements and problem solving methods are introduced.

Systems analysis is the thorough and detailed examination of the real system in order to decompose it into understandable and manageable sized components.

The purpose of the analysis is to establish the interactions, dependencies, and rules governing the components of the system so that a model of the real system can be developed.

Program synthesis is the most creative phase and is subdivided into the modeling and the programming stages. During modeling stage the developer must determine what components are required to model the real system. Components are simplified or even omitted if analysis suggests that their effects do not justify their inclusion (Lewis and Smith, 1979). The task is to keep the program simple, to reduce cost, yet detailed enough to provide reasonable accuracy (Negoita and Ralescu, 1987).

The developer should consider the nature of the system being evaluated, understand the purpose of the simulation, and know what results are expected. The developer can use this criteria to determine the level of accuracy and detail required (Pidd, 1988). There is no need to develop a highly accurate computer simulation if only crude estimates are needed.

During the modeling stage the developer must determine how the program will handle the passage of time, if the model is stochastic or deterministic, and if changes in the model are to be discrete or continuous (Pidd, 1988). As stated above, one of the

advantages of simulation is that the speed at which the experiment takes place can be faster than the real system. Time slicing and next event techniques can be used to handle time-flow within the simulation. Time slicing is a simple technique that controls the flow of time through the simulation in equal time intervals. The next-event technique is used when the system will include slack periods of inactivity. In this technique the model is examined or updated only when a state of change occurs.

The developer must determine if the simulation of the real system is stochastic or deterministic. "A stochastic system is one whose behavior cannot entirely predicted, though some statement may be made about how likely certain events are to occur," (Pidd, 1988). Probability distributions are used in stochastic simulations. As the simulation proceeds, samples from probability distributions are used to determine the stochastic behavior of the model. Deterministic systems are those whose behavior is completely predictable. In deterministic simulations, each time we start with the same initial conditions we get the same results. Deterministic systems are usually expressed as mathematical models.

During the modeling stage the developer must also determine how the variables of the model must

change with respect to time to simulate the real system. In a discrete simulation the variables change only at known or predictable times. In models that allow continuous change, the value of the variables change continuously as the simulation proceeds. These continuous changes could be represented by differential equations and, in theory, their value could be determined at any point in simulation. In reality, digital computers operate with discrete quantities. Changes in variable values cannot change continuously. Continuous change can be simulated by inspecting or changing the value of the variables at a multitude of fixed points in simulated time (Pidd, 1988).

During the programming stage the computer program is designed and the appropriate computer language is selected. The program should be organized with a highly structured approach. Highly structured programs have well-defined subroutines, modules, or procedures of manageable size (Pidd, 1988). Structured programming is always good practice but is particularly important in large and complex programs. Verification of the program is much easier if each module can be tested separately as the program is being built. Structured programming also expedites renovation of the program as the simulation evolves from a thesis to the final model.

In the mid 1970s graphical interface with computers began to replace batch mode operations. Today, users take for granted that data can be entered directly from the keyboard or using some help device such as a mouse (Pidd, 1988). Well designed graphics are the foundation of the user-system interface of the decision support system. Many benefits come from using dynamic graphical displays in simulations. Properly designed graphical displays give the user an idea of the logical behavior of the simulation program. Graphical displays can also aid the user when experimenting with the model. They can aid the user by identifying when an experiment has gone aria and should be terminated if there is no need to make a full run of the simulation. The conditions can be changed and the simulation run again. Finally, graphical displays can be designed to return messages as the program runs that tell the programer that the simulation logic is as it should be. This is particularly important when evaluating how the simulation reacts to state changes in the system.

The next phase of the simulation process is the simulation model verification. Model verification is defined as ensuring that the model behaves (runs) as intended (Cellier, 1982). In other words, each line of program code should do what it is supposed to do.

Structured programming will make the verification phase much easier. Modular, structured programming allows the programmer to test each subroutine as they are coded. In this way each module tested and verified before it is combined with other modules that together make up the submodel.

After all modules have been tested and verified each submodel and the overall model are tested (Cellier, 1982). At this level testing becomes more difficult. Two approaches are commonly used to verify submodels and models. The first is static testing where the computer program is analyzed to determine if it is correct by using correctness proofs, syntactic decompositions, and examination of the structure properties of the program (Cellier, 1982).

The other method is dynamic testing. In dynamic testing the computerized simulation is run under different conditions and the values obtained are used to evaluate the correctness of the program. Dynamic testing techniques include traces, investigation of input-output relations, internal consistency checks, and reprogramming of critical components to determine if the same results are obtained.

In the validation phase confidence in the model is determined by comparing it to the real system to see

how closely the model approximates reality. The purpose of program verification and validation is not to prove that the model runs as intended and is an adequate representation of reality under all sets of conditions (Cellier, 1982). Models are verified and validated over the conditions that are applicable to the realm of operation of the simulation. Designing models that exceed this requirement are uneconomical in time and resources. Many techniques can be used to validate the model. These include evaluation of graphical displays to see if the model appears to be acting correctly, face validity checks, where knowledgeable people determine if the model is reasonable, internal validity, where several runs of stochastic models are made to determine the stochastic variability of the model, and subjecting a sample record from the model to statistical tests to determine correctness of the data.

The final phase of computer model simulation is model analysis. After the computer simulation model has been verified and validated it is ready for computer simulation experiments. Alternate input values and conditions are applied to the model to study the effects on the model output. Model analysis is the culmination the other phases of computer model simulation.

Two Dimensional Finite Element Models

The finite element method was originally developed to evaluate structures. Over the past twenty years it has become an effective tool for evaluating a wide variety of problems in the field of continuum mechanics (Froehlich, 1988). Finite element methods have been successfully used to analyze heat transfer problems, aircraft structural stability and behavior under vibration, friction modeling in structures, the design of warheads and ballistic penetrators (Tipnis and Patton, 1988), and a wide variety of fatigue analysis studies in mechanical engineering. Only in recent years have finite element methods been used to model surface-water flow problems. Several mathematical finite element models have been designed to model two-dimensional surface water flow in a horizontal plane.

The finite element method is a numerical procedure for solving the differential equations encountered in problems of physics and engineering (Froehlich, 1988). Application of this method produces a set of simultaneous nonlinear algebraic equations that must be solved by iteration. Typically there are several thousand equations with several thousand unknowns (Gee, 1985). Models vary in the numerical methods they use to solve these equations.

The common thread in finite element models is that they all require rapid calculations and storage and retrieval of intermediate solution values. Computers are well suited to this task. In the past only large mainframe computers were available. Batch input decks were the norm, and data pre- and post-processing was laborious. Smaller and more affordable computers have opened the door to the use of finite element methods by small companies and individuals who could not afford large and expensive main frame computers, or computing time. Many finite element models designed for mainframe computers have be re-coded and are portable to personal computers (PCs).

Two dimensional surface-water models are sufficient for most practical hydraulic engineering problems, particularly where the horizontal distribution of flow quantities are the main interest. They are best suited for modeling of shallow rivers, flood plains, estuaries, harbors, coastal areas, and almost anywhere the depth-to-width ratio of the body of water is small. In an interesting study a two dimensional finite element model was used to simulate current velocities created by the operation of an existing dam and lock. The concern was that the structure was causing current velocities that created a hazard for small boat operation, and effected the

aquatic habitat downstream (Gee, 1985). They have also been used effectively to model sediment transport and water quality simulations. Probably the greatest strength of these models is that they can simulate flow around and over irregular topography and geometry such as islands and submerged obstacles. In some instances, such as flow over and through a submerged bridge, flow is modeled as a combination one and two- dimensional flows.

Finite element based hydrodynamic models simulate both steady and time dependent two dimensional surface water flow. They solve vertically-integrated equations of motion and continuity, and use the finite element method of analysis to obtain depth-averaged or depth-integrated velocities and flow depths (Froehlich, 1988). All models include the effects of surface roughness of the geometry, as well as fluid stresses caused by turbulence. Most models have the ability to include the Coriolis force and surface wind-stress.

Finite Element Modeling System Operation

The steps taken to apply finite element two dimensional hydrodynamic models are as follows: (1) data collection, (2) network design, (3) calibration, (4) validation, and (5) application (Froehlich, 1988).

This is very similar to the steps in model development discussed earlier.

Data collection. The first step in a surface water modeling is to define the problem system and gather data. Two types of data are collected. Topographic data describes the geometry of the area under study and includes an evaluation of surface roughness to be used in estimating the bed friction coefficients. Hydrologic data consists of evaluation of stage and flow hydrographs, field spot measurements of stage, flow and velocity, rating curves, high-water marks and limits of flooding, and wind measurements. Hydrologic data are used to define the model boundary conditions, and are used later to calibrate and validate the model.

The type and amount of data required depends upon the purpose of the model. It is difficult to establish the minimum data requirements for a particular application. Time, manpower requirements, funding, and the objective of the study will determine the degree of detail in the finite element network design. If the network provides a high level of detail then the risk of not properly representing the system will be reduced. However, if only general approximations of the real system are needed, then it would be uneconomical to collect voluminous data.

Knowledge of the important physical processes that govern the response of a system under study is needed to evaluate the trade-offs between risk of not accurately representing the system and the difficulty of obtaining a solution (Froehlich, 1988).

Network design. The next step is the design of the finite element network. Network design is nothing more than dividing the real system topography into several finite elements. The goal of the network design is to create a finite element network that adequately approximates the real system. Because every network design problem is unique there are no set rules that are applicable in every case. The following are general guidelines that are applicable in most cases.

Design of the work requires deciding the number, size, shape, and configuration of the elements that will approximate the real system. Smaller elements improve the accuracy of the solution but also increase the number of computations. Elements need to be small enough to provide reasonable accuracy, yet large enough to be computationally economical.

The first step is to draw the network on a map of the study area. Because several iterations of the network will probably be constructed it is advisable to draw the network on some type of overlay material, such

as gridded mylar. The grid on the mylar will assist in determining the coordinates of each node, while the map is used to determine their elevation. The scale and detail of the map will determine the accuracy of the model.

The next step is to define the area to be modeled. Model boundaries should be located where the water-surface elevations and flows can be specified accurately. If the exact locations of the water-elevations are unknown or vary, then the boundaries should be placed far away from the areas of primary interest. This will minimize the influence of error introduced by the boundary placement.

After the boundaries have been identified the study area is subdivided into relatively large areas of similar topographic and surface roughness characteristics. Subdivision lines between the regions should follow abrupt changes in topography and surface cover.

Most two dimensional hydrodynamic models will accept 6-node triangular, and 8-node and 9-node quadrangular elements. The two kinds of quadrilateral elements are similar except the 9-node element has an internal node. This additional node requires more computational time but provides greater accuracy than the 8-node element. For most networks a combination of

the three element types will provide the best representation of the study area.

A network of uniformly sized elements is easy to construct but may not necessarily be the best way to model the system. One of the strengths of the finite element method is that the size and shape of the elements can be varied. Small elements should be used in regions where the topography and surface vary considerably. These regions will cause large gradients in the dependent variables of the model. The key is to keep the elements as homogeneous as possible. Large variations of these characteristics in the same element will cause model solution convergence problems.

Large elements can be used in areas where the topography and surface roughness do not vary a great deal, and in areas where approximate solutions are adequate. Transition from large elements to small element should take place gradually. Large elements should not be placed immediately next to small elements.

An important characteristic of element shape is the aspect ratio. The aspect ratio is the ratio of the longest element side length to the smallest. The optimum aspect ratio depends upon the local gradients of the solution variables. The longest side of the element should be aligned in the direction of the

smallest gradient, and likewise, the shortest side should be aligned with the largest gradient of the solution variables.

Calibration. A finite element model is a simplified, discrete representation of a complex and continuous physical flow system (Froehlich, 1988). The three dimensional characteristic of the real system are modeled by two dimensional elements and the flow is assumed to obey differential equations in which several empirical coefficients are used. Model calibration is the process of adjusting the dimensions of the finite elements and the empirical coefficients so the values calculated by the model closely approximate values measured on the real system. The hydrodynamic model is ready for calibration when it produces useful data.

The purpose of model calibration is to obtain an accurate mathematical representation of reality and not to force-fit a poorly constructed model to approximate these values (Froehlich, 1988). Sometimes parameters of poorly constructed models can be adjusted so the model produces results are close approximations of the actual measurements. For example, the finite element network is probably a poor representation of the real system if a good fit of the data can only be accomplished if the Manning roughness coefficient used

is three or four times as large as the original value assumed.

The model is calibrated by systematically adjusting parameters until computed and measured values agree as closely as possible. Measured calibration data consists of spot values of water-surface elevation, flow rates, and velocity components. Sensitivity of computed values to changes in model data should be determined. Small changes in input data that cause significant changes in model results indicate that special care should be taken to obtain the input data.

Surface roughness coefficients have the greatest effect on the model solution. The initial estimate of this empirical coefficient should not have to be change much if the two dimensional network is properly designed. Any changes in the roughness coefficient should be reasonable for the vegetative cover and topographic characteristics of the real reach.

Eddy viscosity coefficients usually do not affect the model solution as much as roughness coefficients do. They have the greatest effect where velocity gradients are large. Increasing eddy viscosity coefficients will cause velocity gradients to be reduced, and the horizontal velocity distribution

will become more uniform (Froehlich, 1988). Reducing eddy viscosity coefficients will cause velocity gradients to increase.

If reasonable adjustment of the roughness and eddy viscosity coefficients do not result in close agreement with measured water *surface elevation*, *flow* rates, and velocities, then model discretization and the accuracy of the data need to be examined.

Validation. As discussed in the section about model development, the validation step establishes confidence in the model. During this step the calibrated model is tested to see if computed values closely agree with measured values not used to calibrate the model. Often it is impossible to validate the model because of insufficient data, since measured values were used to calibrate it.

Application. During this step the model is used to simulate a variety of flow conditions. Application of the model must be made with care, particularly if it is to be used to evaluate conditions far outside the range of it's calibration and validation.

CHAPTER III

RMA-2D

Introduction

RMA-2D is the hydrodynamic numerical model around which the river crossing site simulation and evaluation tool is assembled. RMA-2D is a finite element model used to solve two dimensional (in a horizontal plane) depth averaged shallow water flow equations. The finite element technique produces a set of simultaneous nonlinear algebraic equations that must be solved by iteration. Typically there are several thousand of these equations and several thousand unknowns. The unknowns are the two current velocity components and the depth at each computational node. The model is designed to simulate both steady and non-steady state systems.

RMA-2D was designed for the U.S. Corps of Engineers, Walla Walla District to study flow regimes for the proposed Lower Granite Reservoir. It was developed under contract in 1973 by William P. Norton and Ian P. King of Resources Management Associates. Originally designed for mainframe computers, the

program has undergone considerable modifications since that time and is now portable to an IBM AT computer. Modifications to the model include use of curved isoparametric elements, out of core equation solvers, sophisticated pre- and post-processing routines, and simulation of alternately wet and dry elements during a tidal cycle (King, 1988). The latest modification allows a combination of one and two dimensional elements to permit economical simulation of bays with tributary rivers and complex delta systems where only a small area of the system is actually acting in two dimensional fashion.

Model Applicability

General approximations. RMA-2D simulates one and two dimensional depth averaged hydrodynamic systems with constant water density over the entire area. The dependent variables are water depth and the water velocity in the horizontal plane. Reynolds assumptions of shear stresses in turbulent flow are incorporated as eddy viscosity to approximate turbulent energy losses (Daugherty and Franzini, 1977). Either Chezy or Manning equations can be used to approximate friction losses from the channel surface. Coriolis and surface wind effects can also be approximated.

The governing equations are as follows (Froehlich, 1988):

Momentum:

$$h\frac{\partial u}{\partial t}+uh\frac{\partial u}{\partial x}+vh\frac{\partial u}{\partial y}+gh\frac{\partial a}{\partial x}+gh\frac{\partial h}{\partial x}-\frac{h\epsilon_{xx}}{\rho}\frac{\partial^2 u}{\partial x^2}-\frac{h\epsilon_{xy}}{\rho}\frac{\partial^2 u}{\partial y^2}+Sf_x+\tau_x=0 \quad (1)$$

$$h\frac{\partial v}{\partial t}+uh\frac{\partial v}{\partial x}+vh\frac{\partial v}{\partial y}+gh\frac{\partial a}{\partial y}+gh\frac{\partial h}{\partial y}-\frac{h\epsilon_{yx}}{\rho}\frac{\partial^2 v}{\partial x^2}-\frac{h\epsilon_{yy}}{\rho}\frac{\partial^2 v}{\partial y^2}+Sf_y+\tau_y=0 \quad (2)$$

Continuity:

$$\frac{\partial h}{\partial t}+\frac{\partial(hu)}{\partial x}+\frac{\partial(hv)}{\partial y}=0 \quad (3)$$

where:

u = current velocity in the x direction at time t

v = current velocity in the y direction at time t

h = water depth

a = bottom elevation

Sf_x = non-linear Chezy or Manning bottom friction losses in x direction

Sf_y = non-linear Chezy or Manning bottom friction losses in y direction

τ_x = wind and Coriolis effect in the x direction

τ_y = wind and Coriolis effect in the y direction

$\epsilon_{xx}, \epsilon_{xy}, \epsilon_{yx}, \epsilon_{yy}$ = eddy viscosity coefficients

ρ = density of the water

Two dimensional elements. RMA-2D uses isoparametric quadrilateral and triangular elements to represent the geometry of the reach being modeled. The boundaries of these elements can be either curved or straight. A Galerkin weighted residual approach is used to develop the finite element integrals and Gaussian quadrature is used to evaluate the final integral forms. The basis functions used are bi-quadratic for velocity components and bi-linear for water depth (King, 1988).

The Galerkin finite element method begins by subdividing the physical region into a number of subregions, called elements. An element can be either a triangle or quadrangle and is defined by node points located along its boundary. Each node point is unique, and identified. The values of the dependable variables (velocity components and water depth) are approximated within each element using values defined at the element's node points and a set of interpolation functions. RMA-2D uses quadratic interpolation functions to interpolate depth-averaged velocities. Linear functions are used to interpolate flow depth.

Following this interpolation the method of weighted residuals is applied to the governing differential equations for each element. Approximations of the dependent variables are substituted into the governing equations and usually a

residual is formed because they are not solved exactly. The value of the residuals approach zero when they are multiplied by a weighting function and summed at every point in the solution domain. Galerkin's method requires that the weighting functions are chosen to be the same as the interpolation functions. Because the sum of the weighted residuals approaches zero the finite element equations become integrals. Coefficients of the equations are integrated numerically, and all element equations are assembled to obtain the complete system set of algebraic equations. The complete set of equations is solved simultaneously (Froehlich, 1988).

Equation solution. Depth-averaged flow equations are a coupled system of nonlinear partial-differential equations. As discussed above finite element discretization of the governing partial-differential equations yields a system of non-linear algebraic equations. Solving this non-linear equation system is the most costly aspect (in terms of computer time and memory) of a finite element solution (Froehlich, 1988). In order to optimize computing time and storage space, a symmetric equation system should be solved if possible. Unfortunately, the coefficient matrix that is formed in RMA-2D is nonsymmetric because of the

nonlinear inertia and bottom friction terms in the governing equation.

RMA-2D uses a frontal direct solution scheme in lieu of conventional finite element methods. The solution scheme is designed to minimize core-storage requirements and the number of arithmetic operations needed to solve the system of nonlinear algebraic equations. The frontal solution scheme assembles and eliminates element equations at the same time. As soon as an equation is formed completely it is reduced and eliminated from the "active" coefficient matrix (Froehlich, 1988). It is written into a buffer contained in core memory. When the buffer is full, it is written to an auxiliary storage device. The coefficient matrix is usually never formed in its entirety. At any time the active matrix contains only partially assembled equations or complete equations that have not yet been eliminated. The frontal solution scheme requires rapid storage and retrieval of intermediate values for the three unknowns, the two velocity components and the depth at each node. Minicomputers are well adapted to this process by storing intermediate solutions in large arrays rather than using programmed writes and reads (Gee, 1985).

RMA-2D has a variety of options that can be used to model rather complex systems. As they were not used in this thesis they will only be mentioned here. RMA-2D provides a choice of two element wetting and drying routines can be used to simulate tidal marsh effects. One dimensional element networks can be used to model a complex delta system with many individual channels and junctions, large estuary systems where two dimensional modeling is uneconomical, and to model systems where two dimensional detail is required in some areas, but not over the entire study area. RMA-2D allows control structures and locks to be included in the model. This is essential in modeling not only control structures but also submerged obstacles such as bridges.

Program Organization

RMA-2D is a modeling system not just a single program. There are three components to the RMA-2D modeling system: a preprocessor (RMA-1); the hydrodynamic model (RMA-2D); and two postprocessing programs (VECTOR and CONTUR), that assist in interpreting and displaying the data. The data flow and linkage of the three components can be seen in Figure 3.1.

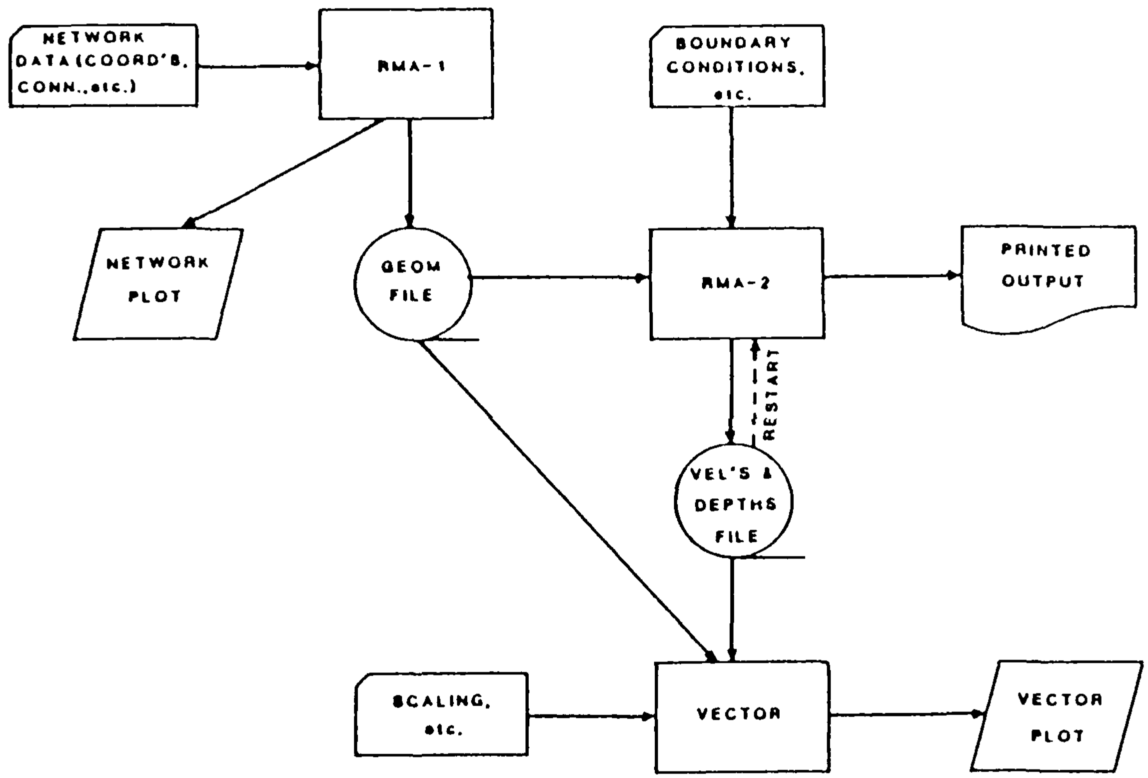


Figure 3.1 RMA-2D program organization

(Source: Gee, 1986, <u>Role of Small Computers in Two-Dimensional Modeling</u>, p. 4)

The data preprocessor, called RMA-1, is an aid in development and error checking of the finite element network. The purpose of this program is to generate two dimensional finite element networks that will be used by RMA-2D. RMA-1 helps the user to develop the element network by generating the quadrilateral and triangular elements. It can also be used to modify and existing network and develop an element order that will allow the most efficient run time for RMA-2D. RMA-1 has the capability to read, edit and print geometric input files from a standard input batch file, or from a file written by a previous run of RMA-1. In addition, RMA-1 can produce a graphical plot of the entire network, as shown in Figure 3.2.

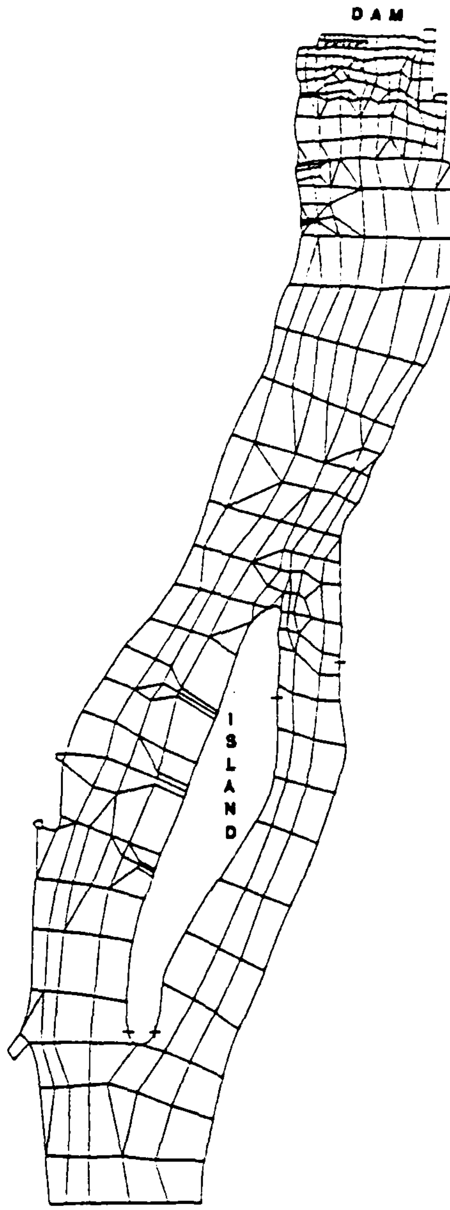


Figure 3.2 RMA-1 finite element network

(Source: Gee, 1986, Use of a Two-Dimensional Flow Model to Quantify Aquatic Habitat, p. 4)

The hydrodynamic model is the second component of the system. Originally designed to use with mainframe computers, RMA-2D still retains some of those attributes. Batch files, in which each line corresponds to a ADP (automated data processing) card are used to input data. The sequence of cards is rigid, as is the spacing of characters and the use of integer and real number values. Output from RMA-2D is in the form of print files that are, unfortunately for most users, in a 132 character format. This too is a hold over from the mainframe computer days.

The postprocessing component of the model consists of two plotting routines that help the user to understand the numerical results of the model run. The first routine, called VECTOR, produces plots of the current velocity calculated in RMA-2D. VECTOR has the capability of plotting either vertically averaged velocity vectors, unit discharge, or near-surface or near-bottom velocity vectors. If the data is from a dynamic simulation, VECTOR can plot these values one step at a time. Additionally, Vector plots can be displayed on the computer screen. Figure 3.3 is a plot of the velocity vectors corresponding to element network in Figure 3.2.

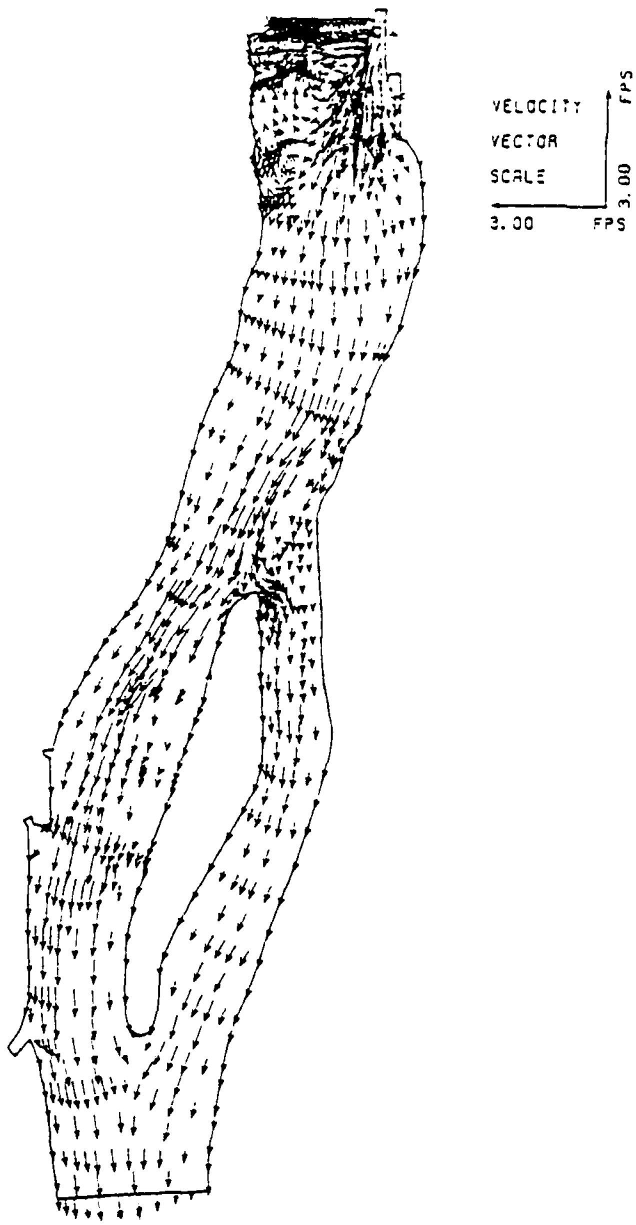
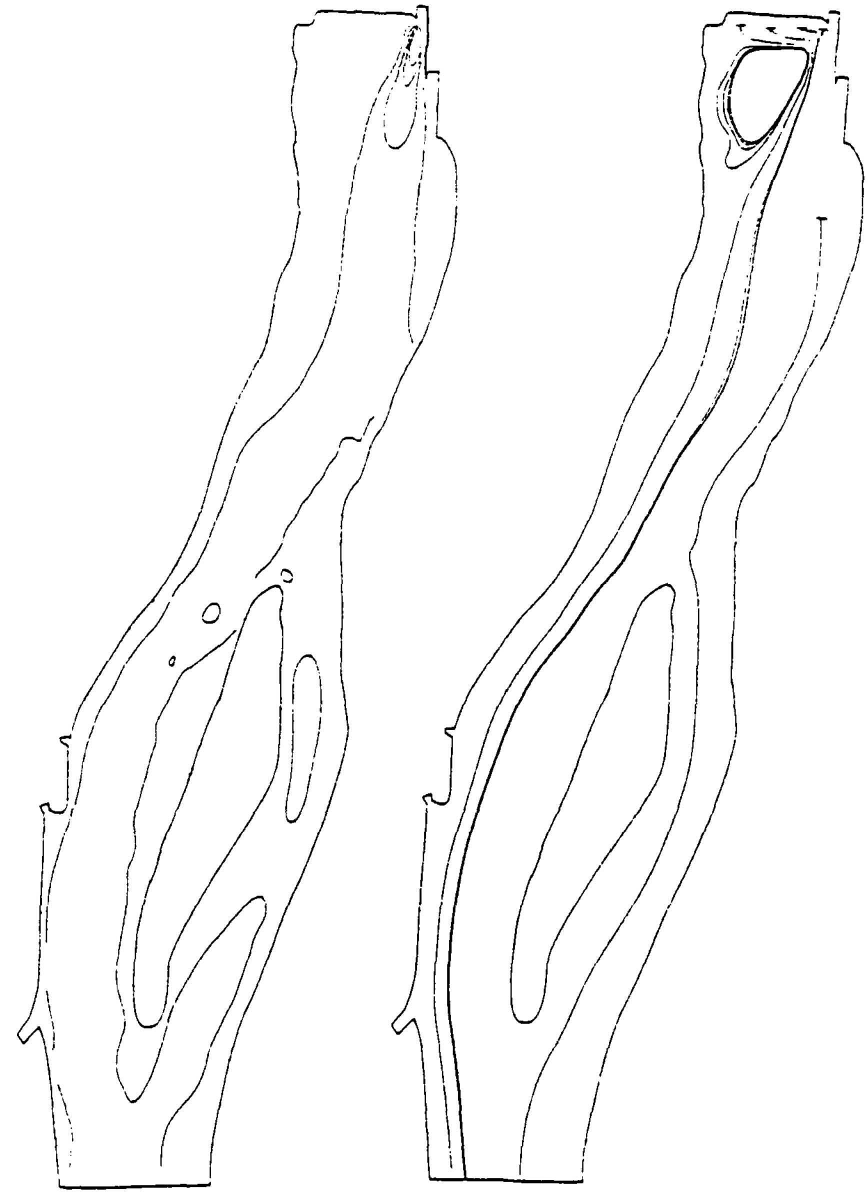


Figure 3.3 RMA-2D VECTOR plot

(Source: Gee, 1986, Use of a Two-Dimensional Flow Model to Quantify Aquatic Habitat, p. 8)

The other postprocessing routine called CONTUR is used to plot contours of data produced by RMA-1 and VECTOR. Data from RMA-1 is used by CONTUR to plot the contours of the bed elevation as approximated in element geometry file. Contur can plot the water surface elevation, isotachs, or bed shear stress from a file generated by the VECTOR routine. The contouring program can also plot the finite element network overlaid on any of the above plots. This is a particularly useful tool in network development and modification. Figure 3.4 is an example of an isotach plot and a pathline plot from the same reach as the previous figures.



Example Isotach Plot Example Pathline Plot

Figure 3.4 RMA-2D CONTUR plots

(Source: Gee, 1986, Use of a Two-Dimensional Flow Model to Quantify Aquatic Habitat, p. 9)

CHAPTER IV

RIVER CROSSING SITE SIMULATION AND EVALUATION TOOL (RC-SET)

General

The River Crossing Site Simulation and Evaluation Tool (RC-SET) is a computer simulation based decision support system used to evaluate potential amphibious river crossing sites. RC-SET allows the user to specify a unique scenario by selecting the crossing site, the vehicle type that will attempt to cross the river, the river flow rate, the vehicle starting point, and the vehicle finish line. It has an interactive graphical interface, prompting the user for needed input, and animating the river crossing attempt in real time as the user actually operates the vehicle.

RC-SET assists the user in interpreting and applying output from the numerical two dimensional finite element hydrodynamic model RMA-2D. The velocity components and water surface elevation values are used by RC-SET to determine the effect of the water current velocity on an amphibious vehicle's translational and rotational movement. The velocity

vectors of the selected flow rate are displayed to help the user select the best route across the reach.

As a true decision support system, RC-SET consists of three major components: the data base management system, the model management system, and the user-system interface (Hopple, 1988). Before raw topographic and hydrologic data can be used by RC-SET, it must be manipulated using a number of data preprocessing tools. The data flow and logical relationship of these components can be seen in Figure 4.1.

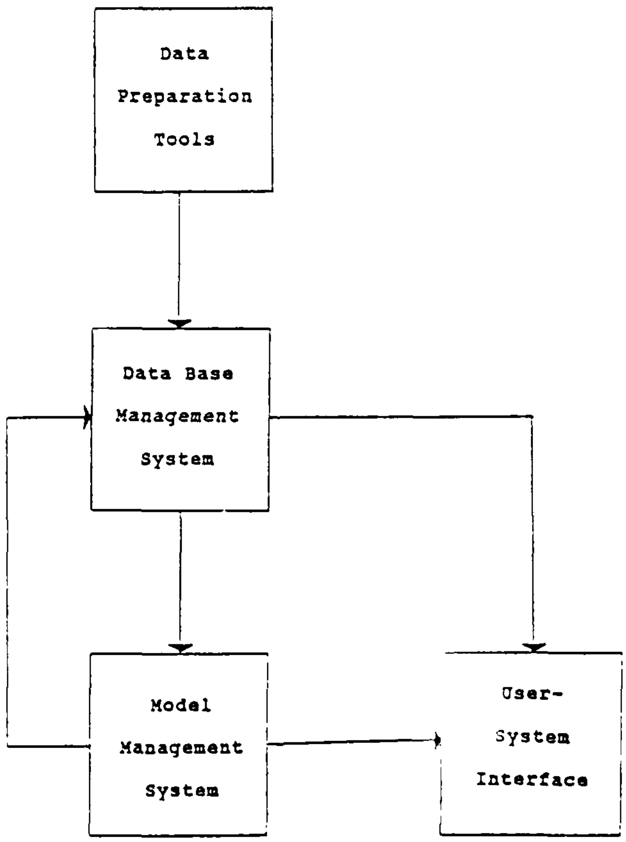


Figure 4.1 RC-SET program organization

External Data Preparation Tools

The purpose of the external data preparation tools is to develop a topography file of the study area and prepare output from RMA-2D for use in RC-SET. The methodology is to first use a raster based geographical information system (GIS) called GRASS to produce a binary image file of the reach topography. This file is used initially by a finite element generating program (MESH) to develop a finite element network of the reach. The same file is used later by RC-SET to display the study area. The output file from the MESH is combined with a batch file defining the study area hydrologic boundary conditions, and together they become the input to RMA-2D. Results from RMA-2D are calibrated with real data from the field when possible. The same procedure is followed for each potential river crossing site to be evaluated using RC-SET. Figure 4.2 shows the flow of data during the preparation of data phase.

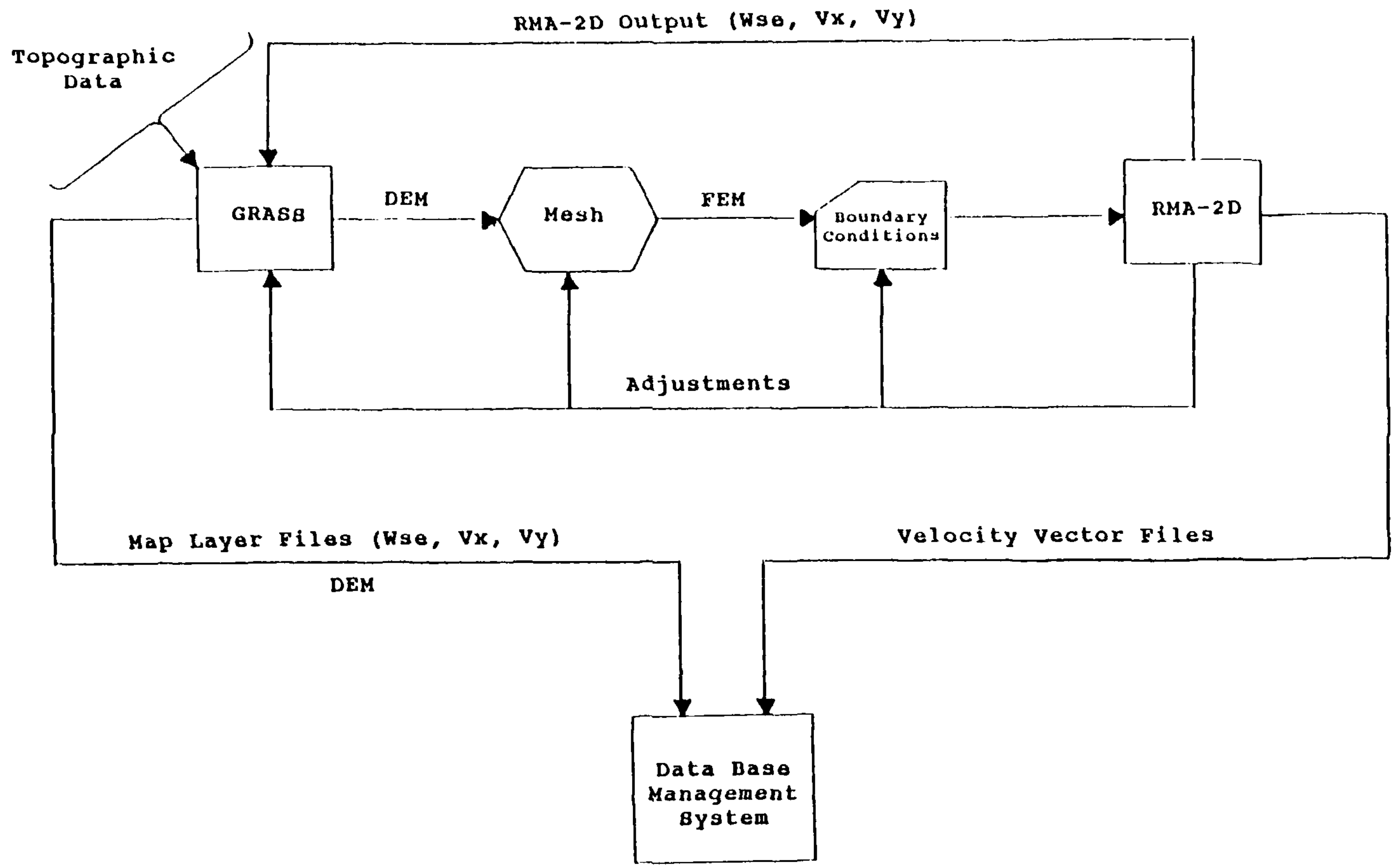


Figure 4.2 Data Preparation Tools

GRASS. The Geographical Resources Analysis Support System (GRASS) is a general purpose, grid-cell based, geographic modeling and analysis program developed by the Corps of Engineers Construction Engineering Research Laboratory (USA-CERL, 1987). It was originally developed for environmental planners at military installations to help them plan land use to minimize the environmental impacts of training and siting of structures. GRASS is now a widely used system, used not only by military planners, but also by many universities, government agencies, and research laboratories. GRASS can be used in studies concerned with environmental planning, facility siting and resource management applications.

GRASS is an interactive graphical program that provides tools for developing, analyzing, and displaying geographical information. It is a grid cell oriented Geographical Information System (GIS). A geographical information system is capable of doing proximity analysis, weighting overlays, and neighborhood processing of geographical data from satellite imagery, paper maps, and other sources (USA-CERL, 1987). GRASS represents the data base as a raster image of continuous rectangles. The analogy used in the GRASS manual is that the map sheets and overlays are checkerboards with each square being

assigned a category number. Each map sheet has some theme such as vegetation, use, or land cover. The category number corresponds to some feature such as the ground surface elevation, a roadway, forest land or even real estate zoning. Anything can be a category. GRASS allows the user to build a data base of up to 50 different map layers. Each map layer is analogous to a separate map sheet.

GRASS includes both image processing and GIS system functions. Image processing includes georeferencing and classification of raw aerial image data from satellite images and high altitude photography (USA-CERL, 1987). It consists of three sub-systems: GRASS-Grid, GRASS-IMAGERY, and GRASS-MAPDEV. The GRASS-GRID is a grid cell analysis subsystem that permits the user to overlay, analyze, manage and display grid cell data bases. GRASS-IMAGERY is a subsystem that extracts and interactively classifies aerial photographs and satellite imagery. Georeferenced images can be transferred to GRASS-GRID for manipulation and display. Finally, GRASS-MAPDEV is a subsystem designed to turn existing hard copy maps into digital displays, and to process digital data from Defense Mapping Agency (DMA), United States Geological Survey (USGS), and other formats.

In the context of this thesis, GRASS is used to produce a geographical model of the study area. The category assigned to each cell is the ground surface elevation. GRASS has the capability of producing complete topographic models from limited data points, but will produce more accurate geographical models from more complete data. GRASS-Grid is used to linearly interpolate between existing cells to fill in any areas where no data is available. The raster image file and a header file, identifying the boundaries of the study area, are imported to the next data preparation tool called MESH.

MESH. MESH is an interactive graphical mesh design system used to generate two dimensional (in the horizontal plane) finite element networks. It was developed at the University of Colorado Center For Advanced Decision Support for Water and Environmental Systems (CADSWES) (Over and Zagona, 1989). MESH is similar to RMA-1 in that it is a preprocessor for two dimensional hydrodynamic mathematical models. However, MESH has several advantages over RMA-1. The primary and overwhelming advantage of MESH is that it provides an interactive environment for the user to generate and modify the finite element network (mesh). Often automatic mesh generation routines like RMA-1 are not

flexible enough to contend with wide variations of velocity gradients and surface features (Over and Zagona, 1989). As stated earlier, calibration of two dimensional hydrodynamic models depends on the accuracy of the finite element network. Inevitably during the calibration process the network will be modified several times until the model produces reasonable results.

MESH reads the binary raster image and header files imported from GRASS and assigns a color to each cell category. The cell map model of the study area is displayed. Colors corresponding to elevation assist the user in interpreting regional characteristics that will effect the velocity gradient. User-system interface takes place via a mouse and menus. As the user selects node locations with the mouse, MESH connects an element side to the node, assigns an identification number to the node, records the node coordinates, and the node elevation. Nodes can be connected to a series node and element sides or separately as arcs. MESH has editing features that enable the user to modify any part of the network at any time. When the user is satisfied with the current network MESH assigns identification numbers to the elements and identifies and calculates the coordinates and elevations of midpoints on each element side. MESH

produces an output ASCII file of the nodal data and element numbers and lists the nodes that define each element. This is the foundation of the RMA-2D batch file input deck.

RMA-2D modeling. The output file from MESH is combined with a batch file defining the hydrologic boundary conditions of the study area to produce the input file to RMA-2D. RMA-2D is calibrated using the same methods discussed in the earlier section about two dimensional modeling. This can be an extremely time consuming task. After the RMA-2D produces results, they should be used to calibrate the model. The first step is to adjust the surface roughness and eddy viscosity coefficients to reasonable limits, then run the model again. If this fails to calibrate the model, then the finite element network is modified using MESH, the boundary conditions reset, and the model run again. The boundary conditions are again adjusted until the model calibrates. If this fails another iteration of finite element network modification is tried, boundary conditions reset and so on. After a number of unsuccessful iterations, it may be necessary to return to GRASS and reevaluate the quality of the DEM. Data used to develop the map sheet are checked for errors, and, perhaps, more data are collected to model the reach correctly.

After RMA-2D is calibrated the output file is modified and ported to GRASS for further map sheet generations. For each water flow rate RMA-2D output files provide, among other things, the node coordinates, depth averaged water velocity in the longitudinal (x) and transverse (y) directions, and water surface elevation at each node. GRASS produces three map layers for each water flow rate: x direction velocity, y direction velocity, and water surface elevation. Map sheet files are converted from binary to ASCII files. The map sheet and corresponding header information files are ported to RC-SET. GRASS also produces cell and header topography files of the study area. A velocity vector data file for each flow rate is also generated from the RMA-2D output file and passed to RC-SET. Table 2 is a summary of the hydrologic and topographic files produce by the data preparation tools and passed to the data base management system.

TABLE 4.1 Data preparation tools output files

Source	File Name	File Type	Use in RC-SET
GRASS	topo.bin	binary	topo graphics display
RMA-2D	vector.dat	ASCII	vector graphic display
GRASS	topo.head	ASCII	topographic data base
GRASS	topo.cell	ASCII	topographic data base
GRASS	wse.head	ASCII	hydrologic data base
GRASS	wse.cell	ASCII	hydrologic data base
GRASS	vx.head	ASCII	hydrologic data base
GRASS	vx.cell	ASCII	hydrologic data base
GRASS	vy.head	ASCII	hydrologic data base
GRASS	vy.cell	ASCII	hydrologic data base

<u>**Data Base Management System**</u>

The data base management system is the first of the three components of the RC-SET decision support system. The purpose of the data base management system is to open and read files selected by the model management system, create topographic and hydrologic data bases from these files, and when prompted, send specific topographic and hydrologic information back to the model management system. The data base management system also has direct file reading subroutines used by the model management system to read topographic binary graphic files and the vehicle data base files.

The data base management system is based upon a similar system used in MESH. Figure 4.3 is a schematic of the data base management system and shows selection and data flow relationships.

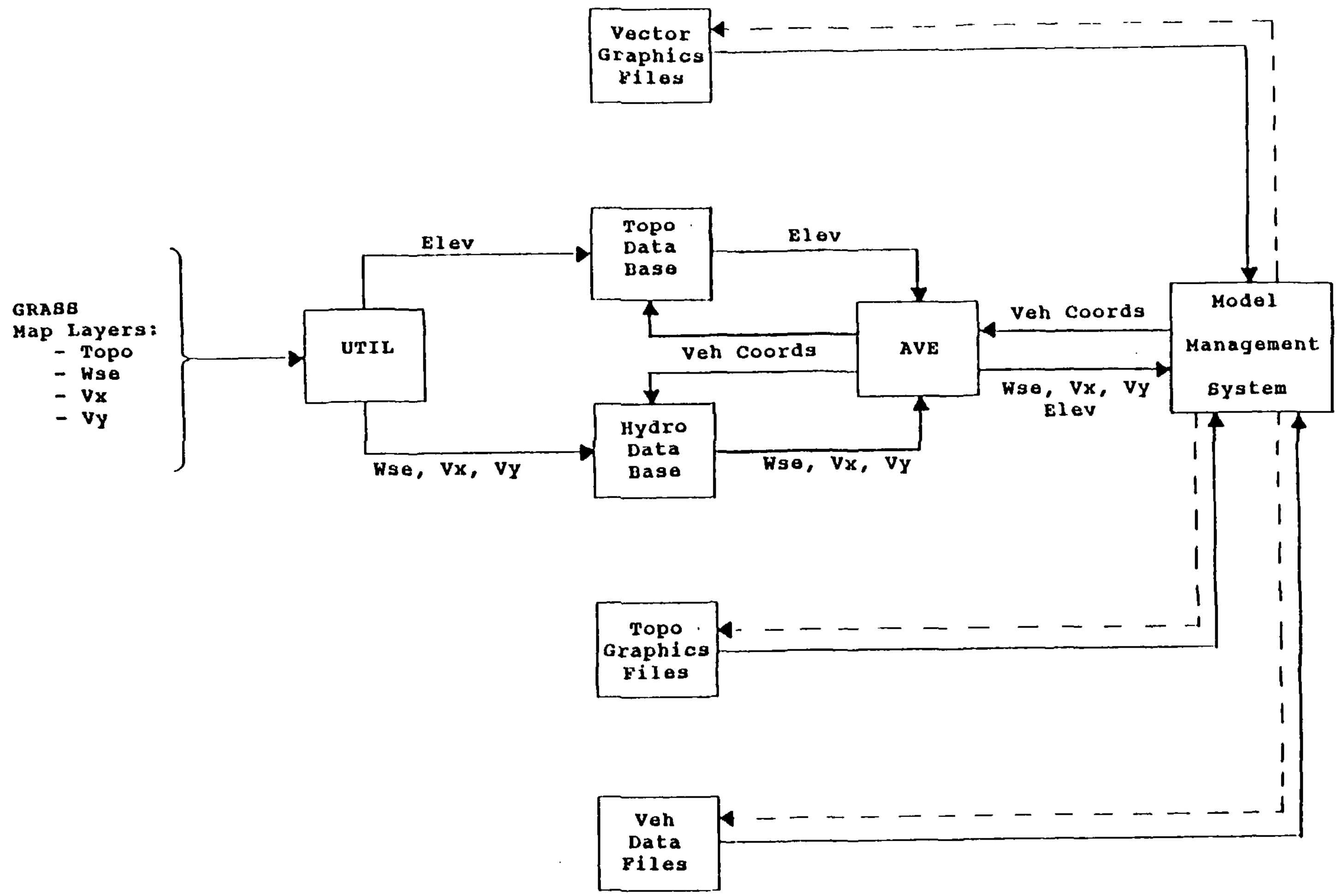


Figure 4.3 Data Base Management System

While Figure 4.3 may look complex, the data base management system is really quite simple. Dashed lines in the figure correspond to topography, vehicle, and flow rate selections from the model management system that indicate which data base files are to be built. Solid lines indicate data flow, either as coordinates from the model management system, or topographic and hydrologic data corresponding to those coordinates being passed back to it.

RC-SET allows the user to build the simulation by selecting the river crossing site, the vehicle type that will attempt to cross the river, and the flow rate of the river. As these selections are made, the data base management system opens and reads the appropriate files and builds the topographic and hydrologic data base. The model management system opens and reads the vehicle data base, and the topography and velocity vector graphics files directly. As the simulation is run the model management system sends the coordinates of the vehicle to the data base management system, which returns corresponding topographic and hydrologic data. The model management system uses this data to calculate the translational and rotational motion of the vehicle, and its new position. The new coordinates are passed to the data base management system and the process continues.

The data base management system uses two computer subroutines to build and access the data base. They are modifications of similar routines used in MESH. The first, called UTIL, is a file utility that reads the header and cell file information from the GRASS ASCII map layer files. The header file is used to dimension a matrix into which the cell file data is read. UTIL calculates x and y vectors that correspond to the world coordinates of the center of each cell. The vector entries correspond to the location where the class information from that cell is stored. The class information is either the ground surface elevation, water surface elevation, water velocity component in the x direction, or the water velocity component in the y direction.

The other computer software subroutine is called AVE is used to access the data base matrices and linearly interpolates the class value at the current vehicle coordinates. UTIL translate world coordinates into vectors that are used to identify a point in space. It reads the class values of the center of surrounding cells and calculates a two-way linear interpolation to determine the class value at the current coordinates. This value is passed back to the model management tool.

Not all topography and hydrologic files are opened and read into the data base. To minimize memory requirements while running RC-SET, only those files corresponding to selections made in the model management system are opened and read into the data base.

Model Management System

The model management system consists of several subroutines that assist the user to set up the model problem and simulate an amphibious vehicle's movement over land and in the water. The user communicates with the model management system via a user-system interface of graphic displays, messages, menus and other devices. The function and design of the user-system interface will be covered in detail later in the thesis. Figure 4.4 shows the logic network of the model management system.

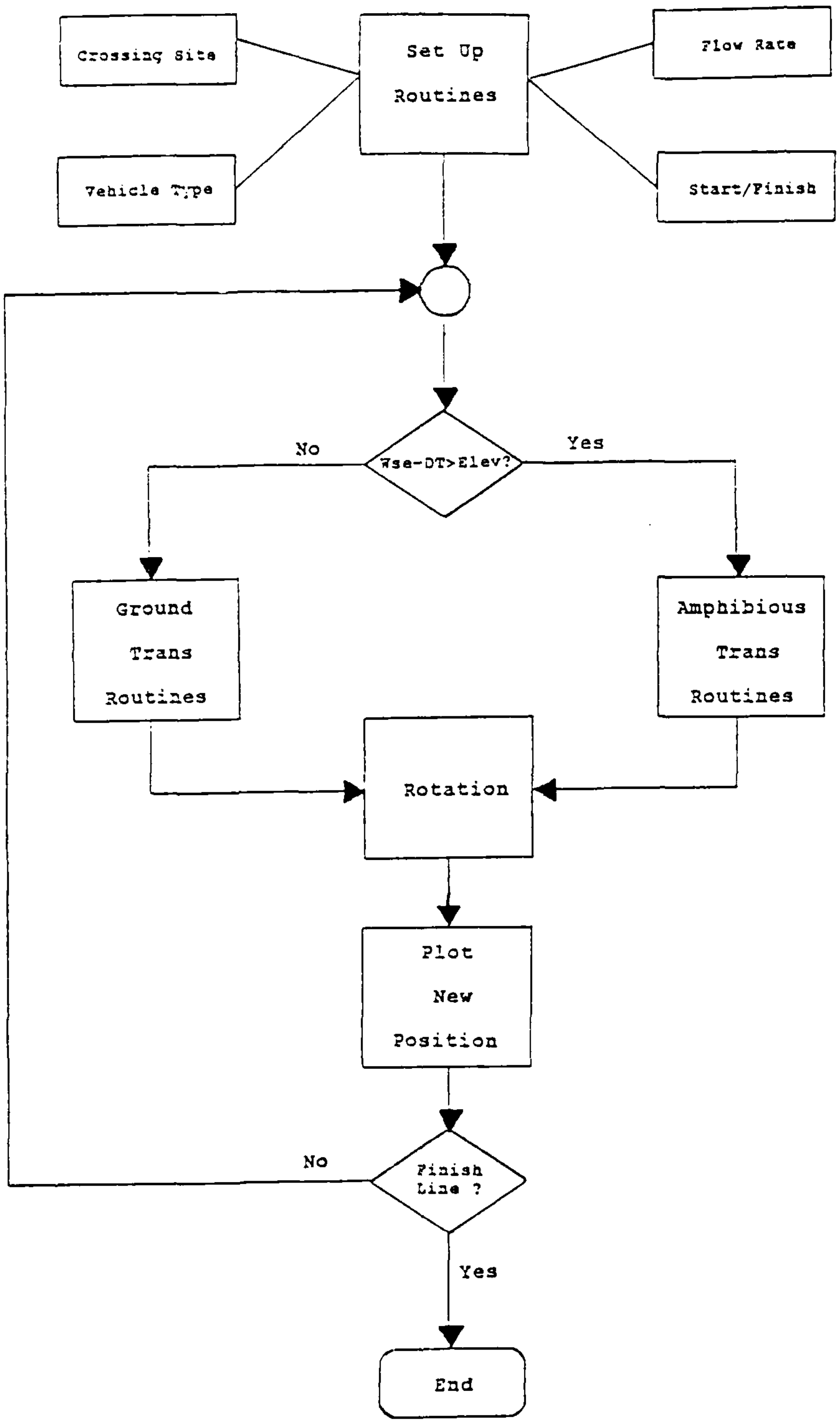


Figure 4.4 Model Management System

Model set up. The model management system allows the user to select the building blocks that make up the simulation model: the crossing site, the vehicle type, the water flow rate, the vehicle starting point, and the vehicle finish line. After the amphibious crossing site is selected from a menu, a raster image of the crossing site is displayed and the topography cell file is open and read into the topographic data base by the data base management system. The user then selects the type vehicle that will attempt to swim across the river. A vehicle data base file is opened and characteristics of the vehicle such as mass, width, height, length, and draft are assigned as global variables.

The user is then prompted to select the flow rate of the water that will flow through the model reach. The velocity vector file is displayed indicating the direction and magnitude of the current velocity at the finite element node coordinates. The data base management system opens and reads the corresponding water surface elevation, and water velocity component map sheet cell files into the hydrology data base.

With the topographic and hydrologic data bases set up, the next step is for the user to select the vehicle starting point. Using the mouse, the user can

select a starting position on either bank or in the water. An image of the vehicle is plotted over the point selected. That point is also the coordinates of the mass center of the vehicle and will be used as the starting point for the movement calculations. Selecting a finish line is the final step in setting up the simulation. The user selects a point on either bank or in the water that becomes the mid point of a forty meter long finish line. A line is used because maneuvering amphibious vehicles in water is very difficult and expecting to exit the river at an exact place is not very practical. When the mass center of the vehicle crosses the finish line the RC-SET simulation will end.

Vehicle movement. RC-SET using a series of modular subroutines to simulate vehicle movement. The users 'drives' the vehicle by using the mouse to manipulate the vehicle steering and acceleration controls. Referring to Figure 9, vehicle calculations are made over a series of five steps. Step one determines if the vehicle will move on the ground or in the water. In step two the translational motion of the vehicle is calculated to determine the vehicle's linear acceleration, velocity, and the world coordinate of the vehicle's mass center. The third step calculates the

rotational motion of the vehicle to determine the angular acceleration, velocity, and angle of rotation. In the fourth step the model management system sends the world coordinates and angle of rotation to the user-system interface for graphic display. The fifth and final step is to determine if the mass center of the vehicle is in the finish line polygon. If it is, the simulation is terminated, if not, the movement routine returns to step 1 and continues as before.

Model limitations. The primary purpose of RC-SET is to simulate amphibious vehicle movement in water. RC-SET also approximates vehicle ground movement, but this feature has not been validated by field data and should be used only as an aid in learning to operate the vehicle or in some other way to enhance the simulation.

Equations determining movement of the vehicle through water have been simplified but still retain a level of accuracy required to accurately approximate the real system. Any body immersed in water will experience forces and moments from the flow. The forces acting on the body in three dimensions are drag, lift and side force, and the corresponding moments are roll, yaw, pitch (White, 1986). Figure 4.5 shows this relationship.

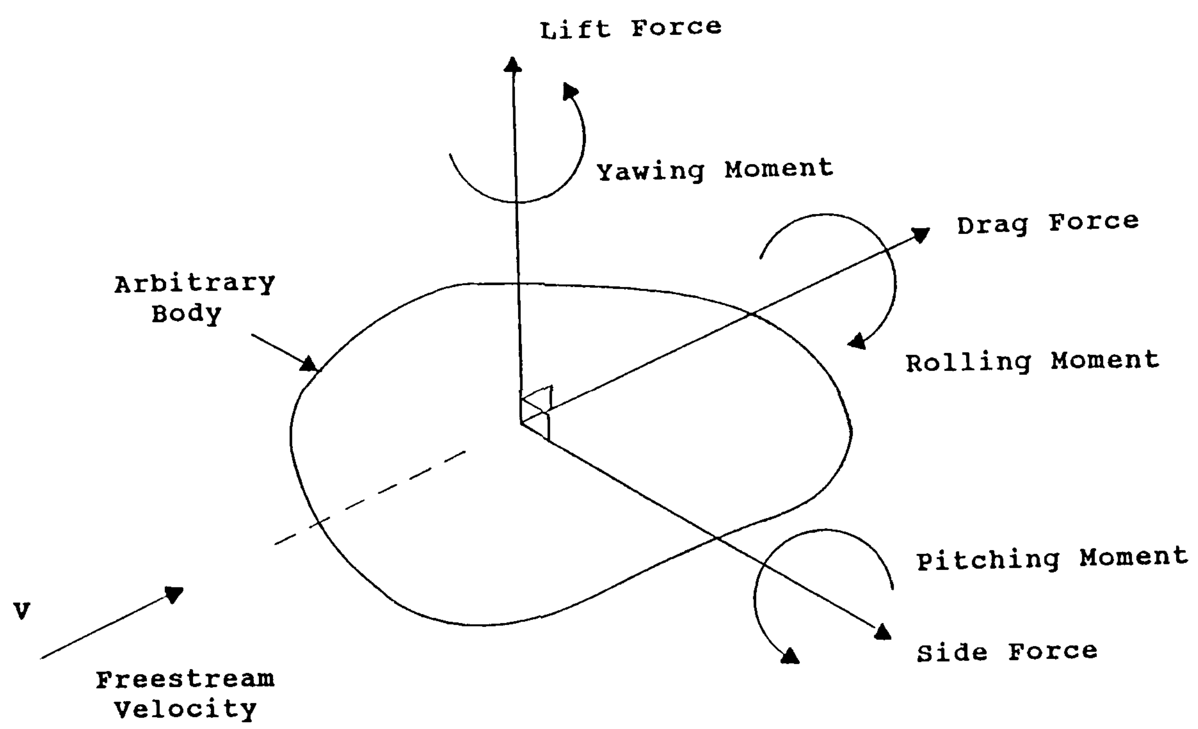


Figure 4.5 Forces and moments on immersed bodies

(Adapted from: White, 1986, <u>Fluid Mechanics</u>, p. 412)

Amphibious vehicle movement in this model is simplified from this three dimensional system to two dimensional system by ignoring the effects of lift, and rolling and pitching moments on the vehicle. Movement of the vehicle in water is simplified to translational and rotational motion in a horizontal plane. In other words, RC-SET determines the vehicles orientation relative to the z axis. This is sufficient to approximate movement of the vehicle as it crosses the river. However, it does not address the danger of the vehicle swamping due to excessive pitching or rolling.

General equations of motion. Specifying the position of a rigid body in the horizontal plane of motion requires the definition of three scalar values, the two coordinates of the mass center and the angular position of the vehicle about the mass center (Meriam, 1978). Three independent scalar equations are required to describe planar motion, one for each scalar value. General plane motion of a rigid body is a combination of translational and rotational motion. The coordinates of the mass center are determined by the equations of translational motion, and the orientation about the mass center is determined by the rotation equation.

Translation is defined as any motion in which every line in the body remains parallel to its original position at all times (Meriam, 1978). In translational motion there is no rotation of the body. In rectilinear translation all points in the body move in parallel straight lines. Figure 4.6 is an example of rectilinear motion in the x and y directions.

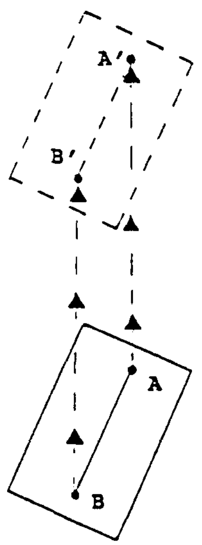


Figure 4.6a

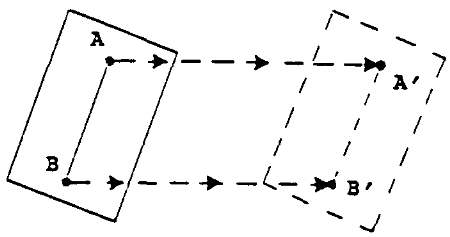


Figure 4.6b

Figure 4.6 Rectilinear motion

(Adapted from: Meriam, 1978, Dynamics, p. 264)

Translational motion is defined by Newton's second law of motion of a mass system. The law states that the resultant of external forces on any system of mass equals the total mass of the system times the acceleration of the system mass:

$$Fx = m(\bar{a}x) \tag{4}$$

$$Fy = m(\bar{a}y) \tag{5}$$

where:

$Fx, \quad Fy$ = the sum of the forces in the x and y directions, respectively

m = the mass of the body

$\bar{a}x, \bar{a}y$ = instantaneous linear acceleration of the body in the x and y directions, respectively

Rotational motion is defined as the orientation of the rigid body about some point of reference. Rotational motion can be calculated using the momentum equation (Meriam, 1978):

$$\sum \bar{M} = \bar{I}\alpha \tag{6}$$

where:

$\sum \bar{M}$ = the sum of the moments about the mass center due to external forces only

$\bar{I}$ = the mass moment of inertia of the vehicle about the z axis

α = the angular acceleration of the vehicle about the mass center, relative to the x axis

Figure 4.7 represents rotational motion about the mass center in a horizontal plane.

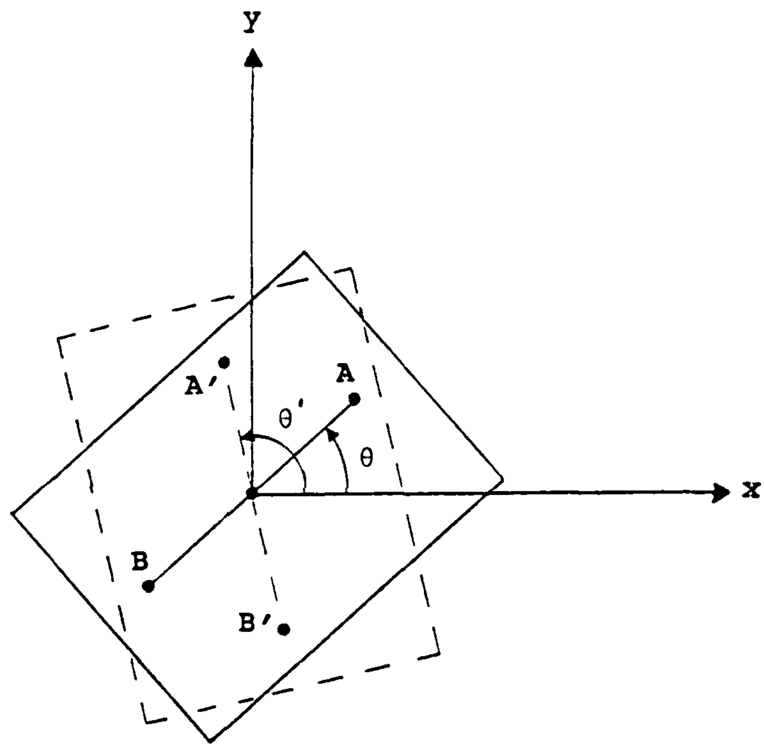


Figure 4.7 Rotational motion in a horizontal plane

(Adapted from: Sears, Zemansky, and Young, 1976, University Physics, p. 160)

The angle θ is the original orientation of the body relative to the x axis, θ' is the new orientation of the body after rotation. The mass moment of inertia is a constant property of the body and is a measure of the distribution of mass around the z axis and through the mass center, and is analogous to mass in translational motion.

Figure 4.8 is a free body diagram describing plane motion (x-y) of an amphibious vehicle moving through water.

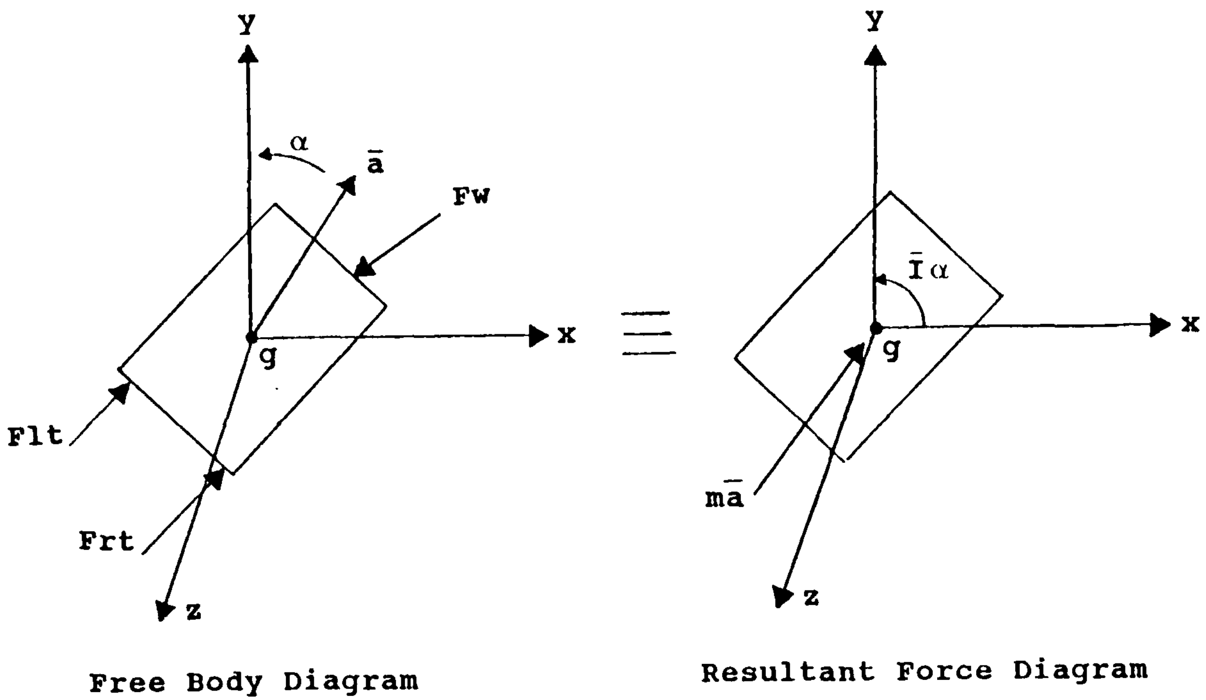


Figure 4.8 Plane motion of an amphibious vehicle in water

(Adapted from: Meriam, 1978, Dynamics, p. 330)

External forces acting on the vehicle are the force of the water, Fw, which include water forces resisting rotation, the propulsive force delivered by the left track, Flt, and the force delivered by the right track, Frt. Acceleration of the mass center is represented by the vector a. Angular acceleration is a scalar quantity and has a magnitude of α. $M\bar{a}$ is the sum of all external forces about the z axis, through the mass center g.

Figure 4.8 also shows the relationship of the free body diagram to the resultant force diagram. The applied forces cause a resultant force ma through the mass center g, in the direction of a, and the resultant couple $\bar{I}\alpha$ in the sense of the angular acceleration α.

As an immersed body moves through water several resistant forces act on it. They are: wave resistance; frictional resistance; form drag; eddy resistance; air resistance; and appendage resistance. Because armored amphibious vehicles are box like, with deep drafts, and move at slow speeds, several of these resistance forces are so small that they can be eliminated from the movement equations. The only significant resistance forces are wave resistance, frictional resistance, and form drag.

Wave resistance is caused by pressure variations around the body that manifest themselves as

elevations and depressions of the water surface (waves). This process upsets the balance of pressures acting on the body which results in a drag force. The magnicude of this drag force is related to the energy of the waves created (Rawson and Tupper, 1968). Amphibious vehicles have haul characteristics that would generate very deep-troughed wave along their sides when traveling at relatively high speeds (Rawson and Tupper, 1968).

The second form of resistance is the frictional force of the water acting against the vehicle. When the vehicle moves through the water, a thin layer of fluid adheres to the surface of the vehicle and has no velocity relative to the vehicle. At some distance from the vehicle, the fluid has some finite velocity. The velocity of the water changes rapidly close to the vehicle surface, but reduces with increasing distance from it. The region where the water velocity changes rapidly is called the boundary layer. The water in the boundary layer is in shear, causing the vehicle to experiences frictional resistance (Rawson and Tupper, 1968).

Water particles moving past the hull in streamlines do not always follow the vehicle form precisely and break away. This, and the presence of the boundary layer effect the pressure distribution

along the hull. The pressure acting on the stern or rear of the vehicle to be reduced, and a corresponding resistive force on the bow or front of the vehicle is produced. This resistive force is called form drag.

The sum of the forces acting on the vehicle moving through the water are as follows:

$$\sum F = Fp - Fw - Ffr - Ffrm \tag{7}$$

where:

$\sum F$ = the sum of the forces acting on the body

Fp = the propulsive force delivered by the vehicle tracks or wheels

Fw = the wave resistance

Ffr = the frictional resistance

Ffrm = the form resistance (pressure only)

Amphibious vehicles are lightly armored tracked or wheeled vehicles capable of negotiating bodies of water under their own power. Nearly all armored vehicles with this capability are designed primarily to transport soldiers and equipment on the ground. Swimming capability is a secondary design concern. Amphibious vehicles use the same means of propulsion on the ground and in the water, e.g. the tracks or wheels. Spinning tracks and wheels are certainly not very efficient propulsive devices in water. As a result

amphibious vehicles in water are typically underpowered and difficult to maneuver.

Because amphibious vehicles move in water at such slow speeds it is difficult to determine which resistive force contributes the most to impede the motion of the vehicle. These resistive forces are combined into a composite resistant force (White, 1986):

$$Fd = \frac{1}{2} Cd\ V^2 \rho A \tag{8}$$

where:

Fd = drag, the total water resistance

Cd = a composite drag coefficient

ρ = water density

V = water velocity relative to the vehicle surface

A = immersed area of the vehicle

The drag coefficient is an empirical value, dependent on the shape and orientation of the vehicle. The relative water velocity is equal to the difference between the water and vehicle velocities (White, 1986):

$$V = v - vw \tag{9}$$

where:

V = the relative water velocity

v = velocity of the vehicle

vw = the water velocity

Translational motion in the x direction can be determined by substituting the components of forces acting in that direction:

$$\sum Fx = Fpx - Fdx = m(ax) \tag{10}$$

where:

$\sum Fx$ = sum of forces in the x direction

Fpx = sum of propulsive forces in the x direction

Fdx = drag forces in the x direction

ax = instantaneous linear acceleration of the vehicle in the x direction

By rearranging terms in equation (10), the instantaneous acceleration can be calculated:

$$ax = \frac{(Fpx - Fdx)}{m} \tag{11}$$

When all forces are applied over a short time period (t), the change in velocity and the resulting change in the resistive force of the water is so small that the water resistive force can be considered as constant over the time period. With this assumption the instantaneous acceleration becomes the constant acceleration during that time period.

After solving for the vehicle acceleration in the x direction it is possible to determine the x component of the vehicle linear velocity and new x coordinate n of the vehicle at the end of time period t (Sears, Zemansky, and Young, 1976):

$$vx = vo + ax(t) \tag{12}$$

where:

vx = the vehicle velocity in the x direction at the end of time period t

vo = the initial vehicle velocity in the x direction at the beginning of time period t

ax = the vehicle acceleration during the time period t

t = the duration of the time period

The x coordinate of the vehicle mass center is (Sears, Zemansky, and Young, 1976):

$$sx = sxo + vox(t) + \frac{1}{2}ax(t)^2 \tag{13}$$

where:

sx = the x coordinate of the mass center of the vehicle at the end of time period t

sxo = the x coordinate at the beginning of time period t

vox = the initial vehicle velocity in the x direction

t = the duration of the time period

ax = the vehicle linear acceleration in the x direction during time period t

The y component of the vehicle's average linear acceleration, velocity and coordinate are calculated the same way.

Figure 4.9 shows the external forces that cause rotation about the mass center of an amphibious vehicle moving through water.

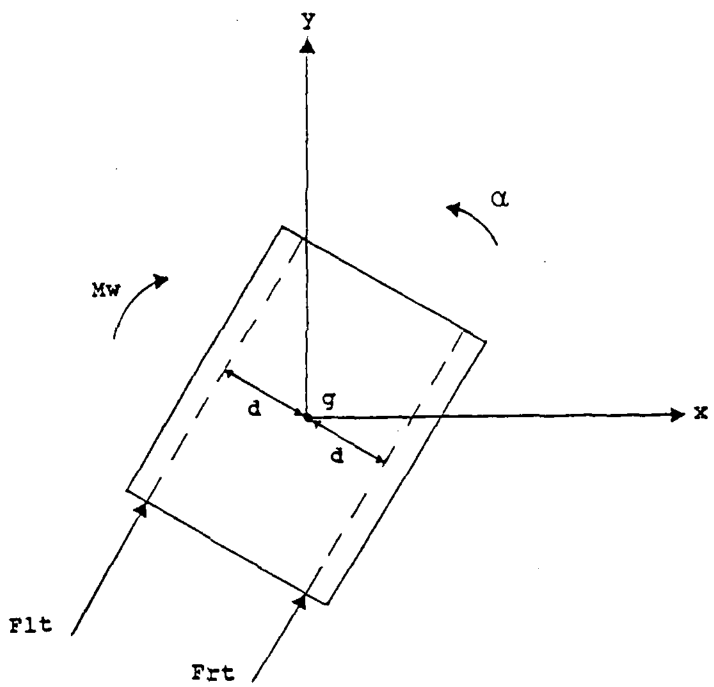


Figure 4.9 Rotational moments about the mass center

The sum of the moments acting about the mass center of the vehicle are:

$$\sum Mg = Flt(d) - Frt(d) - Mw \qquad (14)$$

where:

$\sum Mg$ = the sum of the moments about the mass center

Flt = the propulsive force delivered by the left track or wheels

Frt = the propulsive force delivered by the right track or wheels

d = the moment arm from the mass center to the center line of the left and right tracks or wheels

Mw = the resistive moment of the water acting against rotation

The resistive moment of the water acts against any rotation of the vehicle. The vehicle is treated as a rectangular plate with a submerged area equal to the draft of the vehicle multiplied by the vehicle length. The resistive force of the water acting on the rotating plate is a function of the relative velocity of the water the distance from the mass center to that point. The relative velocity of the plate is greater on that half of the plate rotating counter to the water velocity. Figure 4.10a. shows this relationship.

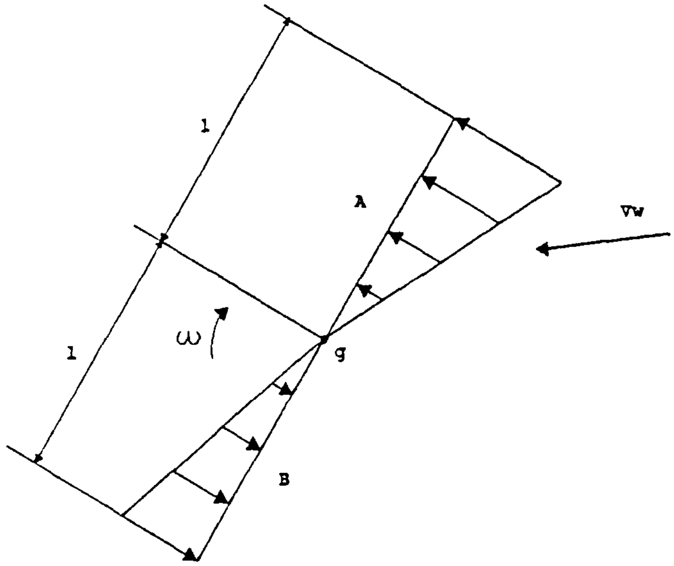


Figure 4.10a

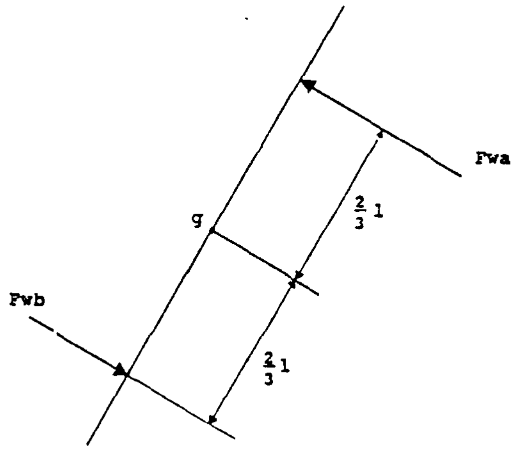


Figure 4.10b

Figure 4.10 Water resistance to rotation

As can be seen in Figure 4.10b., the composite resistive force of the water is the sum of two opposite but unequal forces, Fwa and Fwb, acting two thirds of

the distance from the mass center to the end of the plate. In this respect the plate is analogous to a beam under uniformly varying loads (Timoshenko and Young, 1968).

The moments about the mass center produced by each force is (Timoshenko and Young, 1968):

$$Ma = \frac{(fwa)(l)}{9\sqrt{3}} \tag{15}$$

$$Mb = \frac{(fwb)(l)}{9\sqrt{3}} \tag{16}$$

where:

Ma,Mb = the maximum moments about the mass center

fwa,fwb = the maximum resistive force of the water acting at the ends of sections A and B

The maximum force of the water on section A is (White, 1986):

$$fa = \frac{1}{2}cd(\rho)(v)^2(a) \tag{17}$$

where:

fa = the maximum resistive force on section A

ρ = the water density

cd = the drag coefficient of the plate

a = the submerged area of the plate

v = the relative velocity of the water at the end of section A

The relative velocity of the water at the end of section A is (White, 1986):

$$v = vwn - vo \qquad (18)$$

where:

v = the relative velocity of the water

vwn = the component of the water velocity normal to the plate surface

vo = the maximum velocity of the plate due to the angular acceleration

The maximum velocity of the plate at the end of section A is(Sears, Zemansky, and Young, 1976):

$$vo = \omega(l) \qquad (19)$$

where:

vo = the maximum velocity

ω = the angular velocity of the plate

l = the distance from the mass center to the end of section A

The maximum force acting on section B is determined in the same fashion. The composite moment due to the water resistance on sections A and B is:

$$Mw = Ma + Mb \tag{20}$$

where:

Mw = the moment about the mass center due to water resistance

Ma = the moment about the mass center due to water resistance on section A

Mb = the moment about the mass center due to water resistance on section B

Knowing the moments about the mass center, terms in equation (6) are arranged and the angular acceleration over time period t is determined:

$$\alpha = \frac{Mg}{I} \tag{21}$$

where:

α = the angular acceleration of the vehicle about the mass center, relative to the x axis

Mg = the summation of the moments of all external forces about the mass center

I = the mass moment of inertia of the vehicle about the z axis

The angular velocity of the vehicle is (Sears, Zemansky, and Young, 1976):

$$\omega = \omega_0 + \alpha(t) \tag{22}$$

where:

ω_0 = the angular velocity of the vehicle at the beginning of time period t

ω = the angular velocity of the vehicle at the end of time period t

α = the angular acceleration of the vehicle over time period t

t = the time period over which constant force is applied

And finally, the orientation of the vehicle relative to the x axis is (Sears, Zemansky, and Young, 1976):

$$\theta = \theta_0 + \omega_0(t) + \frac{1}{2}\alpha(t) \tag{23}$$

where:

θ = the angle of orientation relative to the x axis at the end of time period t

θ_0 = the angle of orientation at the start of time period t

ω_0 = the angular velocity at the beginning of time period t

α = the constant angular acceleration over time period t

t = the time period

Movement simulation. Referring to Figure 4.4, the first step of the vehicle movement simulation is to determine if the vehicle will move on the ground or in the water. The model management system sends the present world coordinates of the vehicle mass center to the data base management system. The data base management system accesses the hydrologic and topographic data base and returns the water surface elevation and ground surface at that point. If the elevation at the bottom of the vehicle (equal to the water surface elevation minus the vehicle draft) is greater than the surface elevation, then the vehicle is floating and will move as an amphibious vehicle. If not, the vehicle is on firm ground and will move in the ground vehicle mode. RC-SET assumes that any contact with the ground indicates one hundred percent traction. The simulation is simplified because it does not consider the effects of buoyant forces acting on the vehicle.

Translational motion. The next step is to calculate the translational motion of the vehicle. If the vehicle is floating in the water then the water movement routines are called. The first routine calculates the rectilinear motion component in the x direction. The user-system interface determines

vehicle control information from the vehicle controls on the screen. The gear selection determines if the vehicle will be travelling in reverse, forward, or park. When in park and in the water the vehicle will slow down until the vehicle assumes the speed of the water current.

If the vehicle is in forward or reverse, the routine opens the vehicle data base file and reads the maximum vehicle water speed in that direction, and the associated drag coefficient. These are include in a rearrangement of the translation motion equation to determine the propulsive force required to move at the maximum speed. The propulsive force is the maximum propulsive force that the vehicle can deliver.

The accelerator slider information is used to determine how much of the maximum propulsive force will be used. Steering slider information is used to determine what percentage of this force will be applied to the left and right tracks. The vehicle world coordinates are passed to the data base management system and the water velocity in the x direction is returned. This and the vehicle characteristics from the vehicle data base are used to determine the force of the water acting on the vehicle. The propulsive and water forces are applied to the motion equation for rectilinear motion and the new linear acceleration and

velocity in the x direction, and the new x coordinate are calculated.

A similar routine calculates the same information for the y components.

Rotational motion. Step three of the model simulation is to determine the rotation movement of the vehicle during the current time period. The composite water velocity vector at the current mass center coordinates is used to determine the magnitude and sign of the moment that resists rotation of the vehicle. Propulsive forces in each track are known from the previous steps. These forces and moments are entered into the rotational motion momentum equation to determine the constant angular acceleration during the time period. The angular velocity and angle of rotation, also called the angle of orientation, are also calculated.

In the fourth step the world coordinates of the new position of the mass center and the angle of rotation are passed to the user-system interface. The user-system interface translates the world coordinates into machine coordinates and displays a correctly oriented image of the vehicle.

The fifth step of the movement simulation is to determine if the mass center of the vehicle has crossed

the finish line. Actually, the user-system interface determines if the mass center of the vehicle is inside a polygon defining the finish line. If the vehicle is not inside the polygon then the movement simulation returns to step 1 and continues. If the vehicle crosses the finish line or leaves the model area the movement simulation ends.

User-System Interface

The user-system interface is the link between the user and RC-SET. The user-system interface is a series of menu driven graphic displays used to set up the model, and animated movement graphics used to run the simulation. The foundation of the graphic subroutines is the CADSWES computer graphics Toolbox. Toolbox is a series of graphics tools that allow engineers with limited computer graphics experience to build sophisticated user-system interfaces.

Figure 4.11 is the title screen.

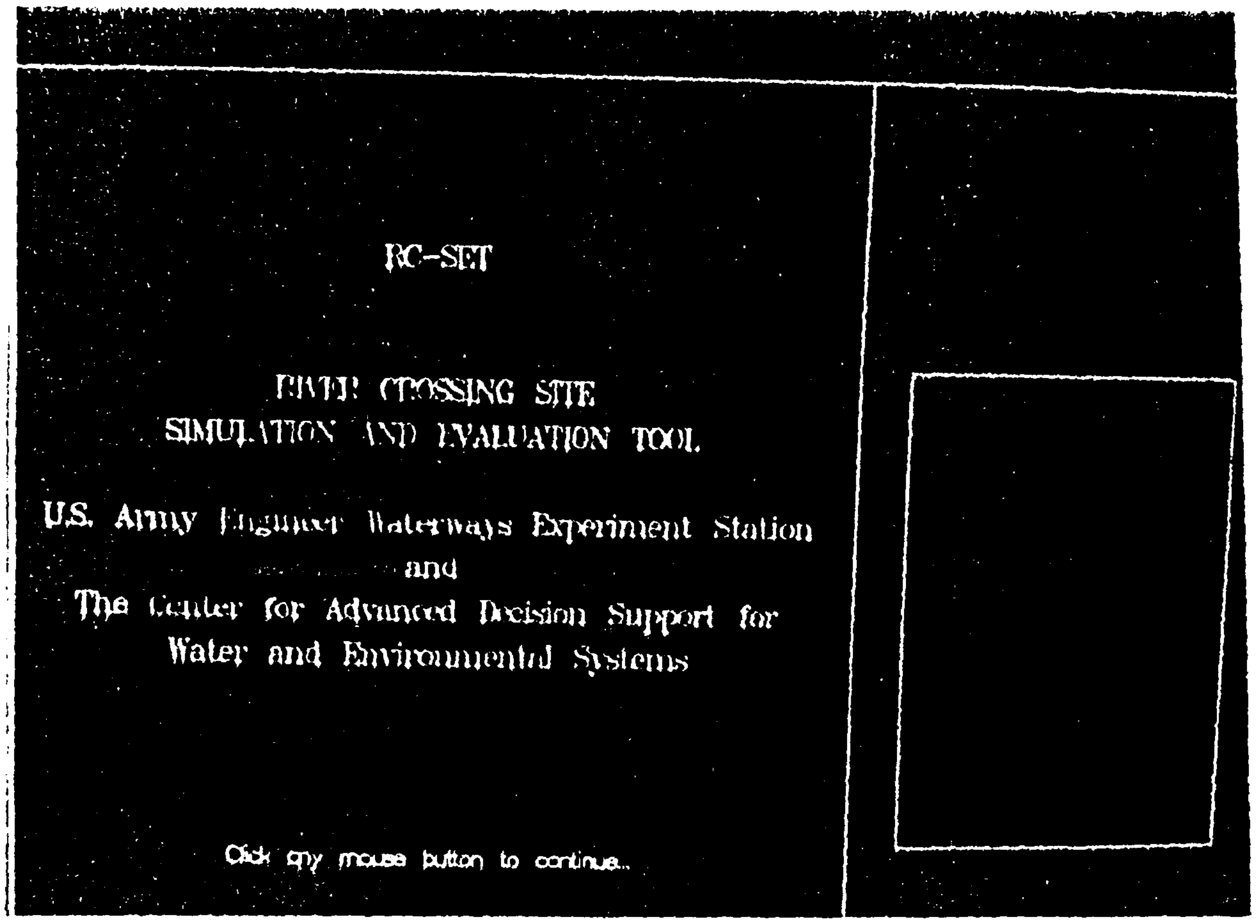


Figure 4.11 RC-SET title screen

The next screen describes the purpose, use and capabilities of the RC-SET decision support system. This is shown in Figure 4.12.

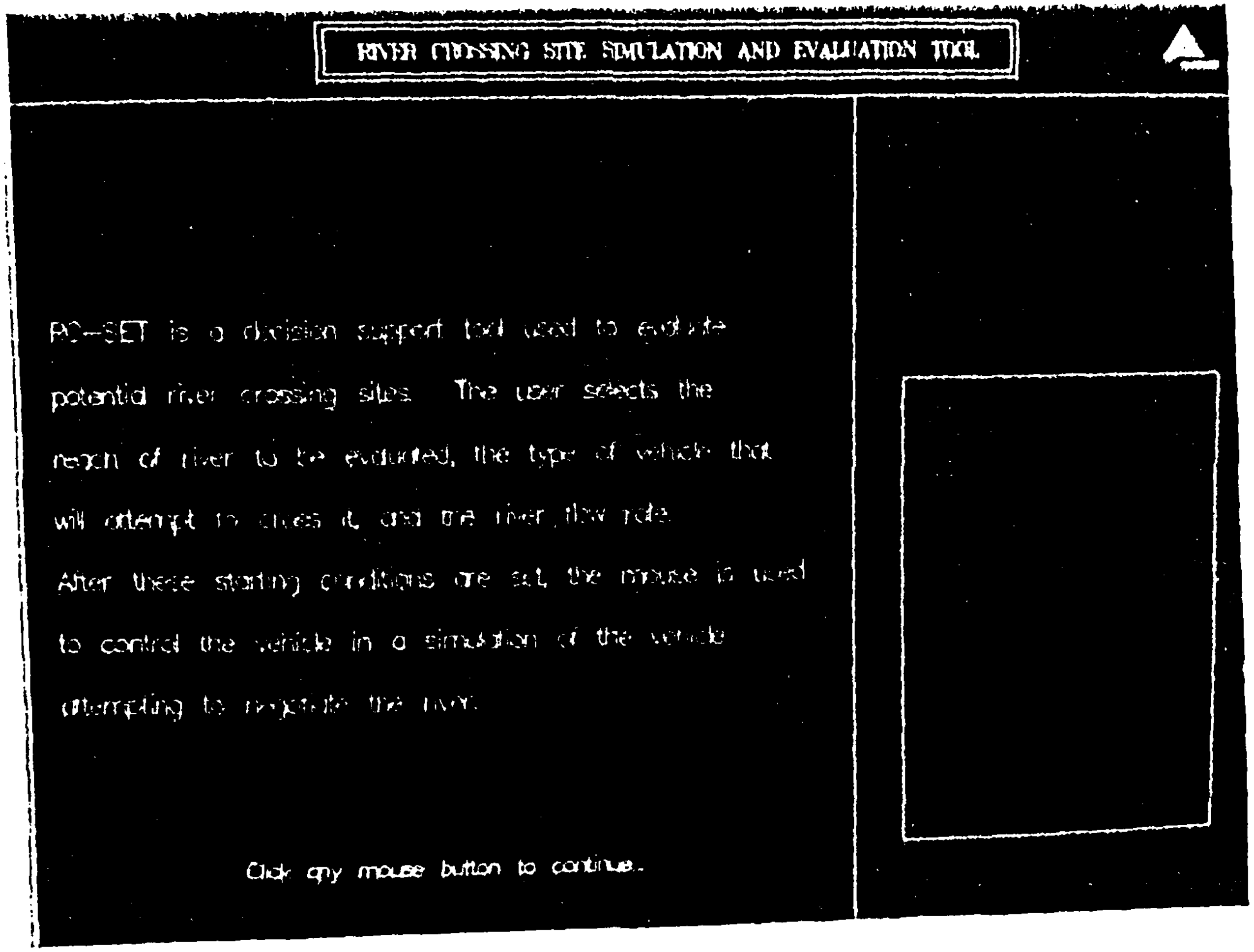


Figure 4.12 RC-SET capabilities screen

Following the capabilities screen the user is prompted to select the river crossing site location from a menu using a mouse. The crossing sites listed in the menu correspond to files that will be read into the hydrologic and topographic data base. The sites listed would typically represent known crossing sites on rivers of tactical and operational significance. After the crossing site is selected a raster image of the reach is displayed on the screen. An arrow indicating the direction of water flow and a north seeking arrow are drawn to help orient the user. A scale is also drawn.

A status table is drawn above the crossing site raster image to help the user remember the river crossing site, vehicle type, and flow rate selected for the simulation.

A menu listing a selection of amphibious vehicles replaces the crossing site selection menu. Each vehicle type on the menu has a corresponding data base file. The user selects a vehicle type with the mouse and a vehicle fact sheet is displayed. The status table is updated with the type vehicle selected.

The vehicle selection menu is replaced by the flow rate selection menu. The flow rates listed on the menu correspond to the hydrologic map sheets produced by RMA-2D and GRASS. After the user selects a flow

rate a vector image of the velocity vectors is overlaid onto the reach raster image. Each velocity vector point of origin corresponds to a node coordinates of the finite element network. The length of each vector corresponds to the magnitude of the velocity at that point. This vector image is one of greatest advantages of applying graphics to two dimensional hydrodynamic modeling because it helps the user to select the best route across the reach. The status table is updated with the flow rate selected.

The user is then prompted to select the starting point of the vehicle. A plan view image of the vehicle is drawn to scale over the point selected. The starting point can be on either bank or in the water, but must be within the boundaries of the raster image of the reach. The user is prompted and selects the finish point. The world coordinates of this point becomes the centroid of a scaled forty by three meter polygon which is drawn on the raster image. The finish line is the goal to which the user will move the vehicle. The simulation will end when the mass center of the vehicle crosses any boundary of the finish line polygon.

The vehicle fact sheet table is replaced with the vehicle control panel. The user-system interface

uses several gages, sliders, and buttons to permit the user to 'drive' the vehicle during the simulation.

The user can determine the vehicles heading and speed, and the elapsed simulation time from the gages. The user steers the vehicle by moving the bar small vertical bar in the steering slider with the mouse. Several small boxes in the steering slider help orient the user to the central or straight forward position. At the extreme right and left of the vehicle control slider are the pivot left and right positions. By clicking on these boxes the user causes the vehicle tracks to spin in opposite directions, thus causing the vehicle to pivot.

The accelerator slider is analogous to the accelerator pedal in an automobile. The user determines how much power will be applied to the vehicle tracks by clicking inside the slider. The red bar inside the slider is a graphical representation of the percentage of force applied.

At the bottom of the vehicle control panel are three buttons that represent the automatic gear shift of the vehicle. The user selects either Forward, Park, or Reverse. After the simulation begins the user can change gears from forward to reverse instantaneously but the vehicle will come to a complete stop before moving in the new direction.

As the vehicle control panel appears on the screen, the flow rate selection menu is replaced by the simulation control menu. This menu allows the user to start, stop, quit, freeze, or exit the simulation.

The simulation will not begin until the user selects Start from the menu. Selecting Pause does not effect the simulation other than to freeze all activity. When the user starts the simulation the user-system interface reads the gear, acceleration, and steering input devices once every second, and sends the status to the model management system. One of the most critical aspects of amphibious vehicle crossings is to minimize the time the vehicle is in the water and exposed to enemy fire. This is why a timer is provided on the vehicle control panel. The timer starts when the simulation begins.

During each movement cycle the new vehicle position and orientation data is received from the model management system. The user-system interface plots a different colored image of the vehicle over the old position, then plots a correctly oriented image of the vehicle at the new coordinates. The result of this is that the vehicle appears to be leaving a trail as it moves. The trail is loam colored during movement on land, and off-white for movement in water. This

CHAPTER V

THE CASE STUDY

Scope

The case study was conducted with two objectives in mind. The first objective was to validate RC-SET as a computer simulation based decision support system. As the data preparation tools and RC-SET components were assembled, each subsystem was tested to verify that it did what it was designed to do. Validating RC-SET will determine if they work together to closely approximate the real system. The second objective of the case study was to determine if RC-SET could be used to evaluate potential river crossing sites. Accomplishing these objectives confirms the thesis that a decision support system can be developed with the capability of using two dimensional hydrodynamic model data to evaluate amphibious river crossing sites.

The challenge of this case study was to find a combination of reach, flow rate data for that reach, and vehicle type that would provide enough data to calibrate, and if possible, validate RC-SET. To

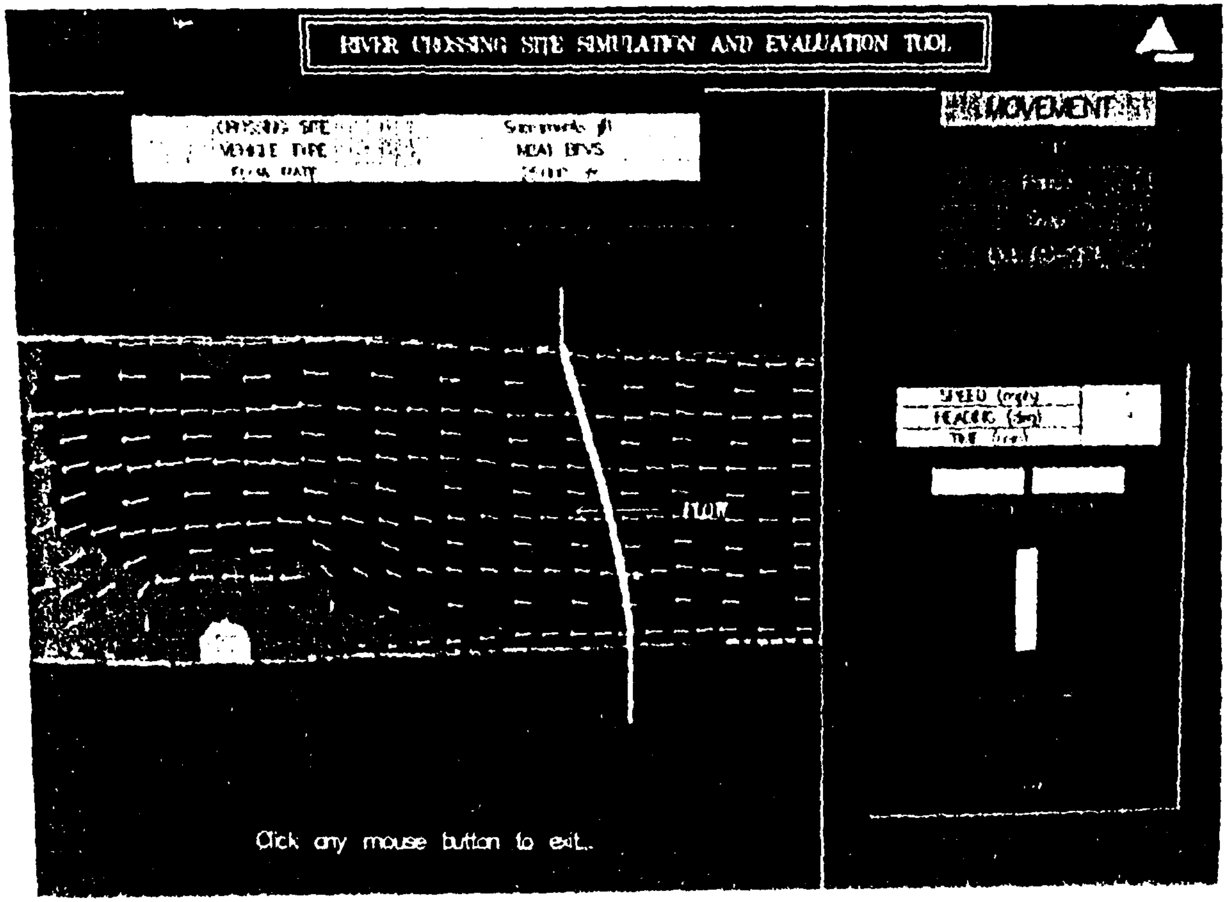


Figure 4.13 A successful river crossing using RC-SET

correctly validate RC-SET, field experiments would have to conducted to determine how a selected vehicle type actually negotiates the flow through the selected reach of river. However, these field experiments have not been conducted. Data from two unrelated field studies, one on a reach and the other on a vehicle type, were used together form an ersatz 'real' system. Complete validation of RC-SET is not possible in this case study because of this lack of field data from the ersatz system. However, there is enough data from the separate studies to calibrate some components of the data preparation tools and RC-SET. In this case study, partial confidence in RC-SET can only be established through limited calibration of some components, and subjective validation, such as reasonability tests. The purpose of this thesis is to develop a prototype decision support system, and in this respect, complete validation is not required, nor is it a realistic goal.

Reach Selection

The existence of sufficient hydrologic and topographic data was not the only criteria used to evaluate candidate river reaches in this case study. Because RC-SET is a prototype, the reach of river should have relatively simple topographical characteristics to facilitate calibration of RMA-2D.

Topographic data was also recorded at each cross section. Cross sections were space at intervals of over four hundred feet. Topographic data from the University of California study was used as input to GRASS to construct a DEM of the reach. Data from seven cross sections were used, totaling ninety-six ground surface elevation data points. With so few data points it was difficult to use GRASS to model correctly the reach topography. After several unsuccessful attempts, a workable method was discovered. Initially, GRASS was used to interpolate between data points and fill in any empty regions.

The absence of data between cross sections caused considerable distortion in the DEM. To smooth this distortion, the channel bottom and overbanks were each modeled separately using selected data points. Strips of data were cut from the overbanks and channel bottom and an interpolation of the slopes was produced. Finally, all section map sheets were pasted together to form the DEM of the reach. Figure 5.2 is a simplification of the DEM construction process using GRASS.

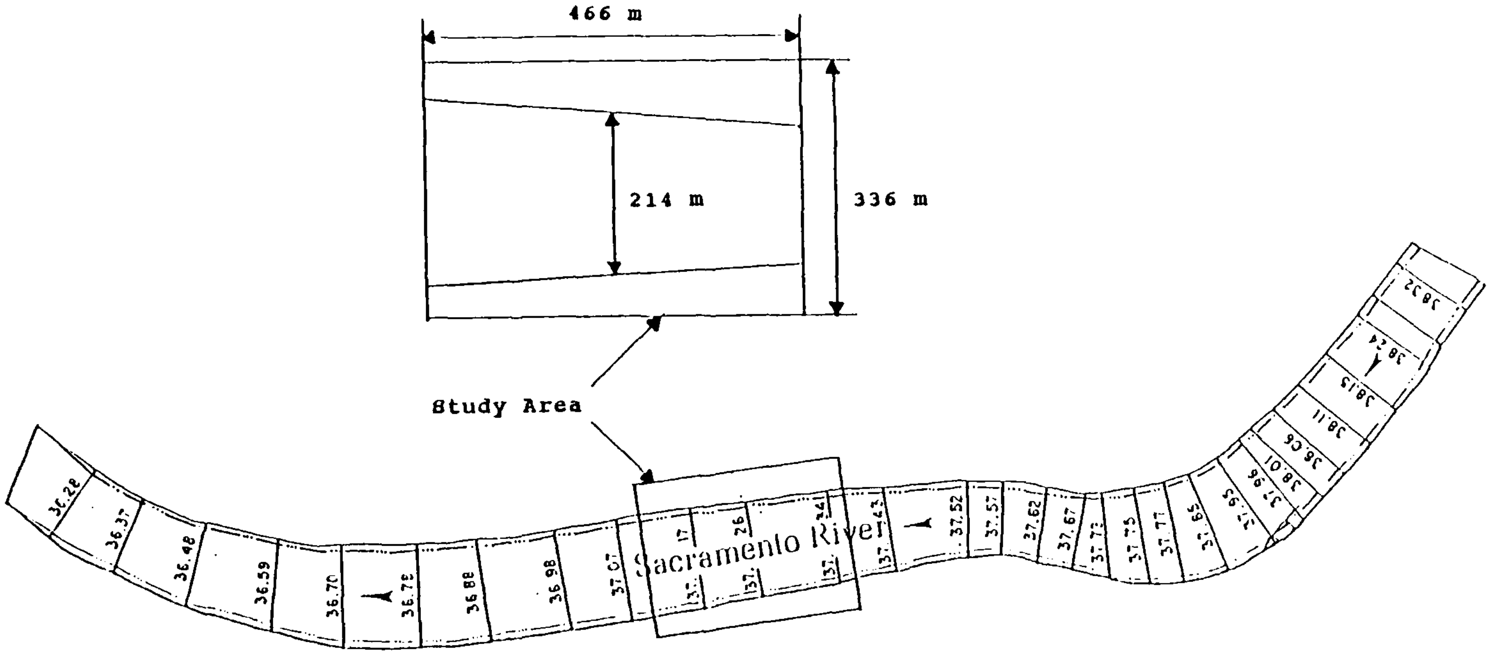


Figure 5.1 Sacramento river study area

(Adapted from: Tod and DeVries, 1987, Physical Model Investigations of River Control Structures, Final Report, p. 3)

However, the topography of the river should be complex enough to cause non-linear current velocities. The maximum velocity of the current should be less than or equal to five feet per second to allow amphibious vehicle maneuvering.

The reach of river selected was a five hundred meter length of the Sacramento River. This reach was the subject of a physical modeling study that evaluated the hydraulic characteristics of various river control structures. The study was conducted by the University of California, Davis under contract from the State of California Department of Water Resources (Todd and DeVries, 1987). Field data for the study was collected at several cross sections of the river and included water surface elevations and depth averaged current velocities at various flow rates. Data from that study was used in this thesis to set the boundary conditions and calibrate RMA-2D. Figure 5.1 is a plan view of the part of the Sacramento river used in the University of California study. The area bounded by the box is the portion of this reach used in this case study.

trail is a valuable tool when used to evaluate the route taken at the completion of the simulation.

The simulation can be ended in a number of ways. The user can terminate the RC-SET program at anytime during the model building and simulation phases from any of the menus. This gives the user the flexibility to terminate simulations that are obvious failures or are incorrectly set up. The user can also stop the simulation from the simulation menu. The simulation will stop at that point but will continue to be displayed until any mouse button is clicked. The program will also stop if the vehicle moves outside the boundaries of the reach image. If this happens the reach will be displayed until Quit RC-SET is selected from the simulation menu. The last way to stop the simulation is to successfully maneuver the vehicle until it crosses into the finish line polygon. Figure 4.13 is a successful amphibious river crossing simulation using RC-SET.

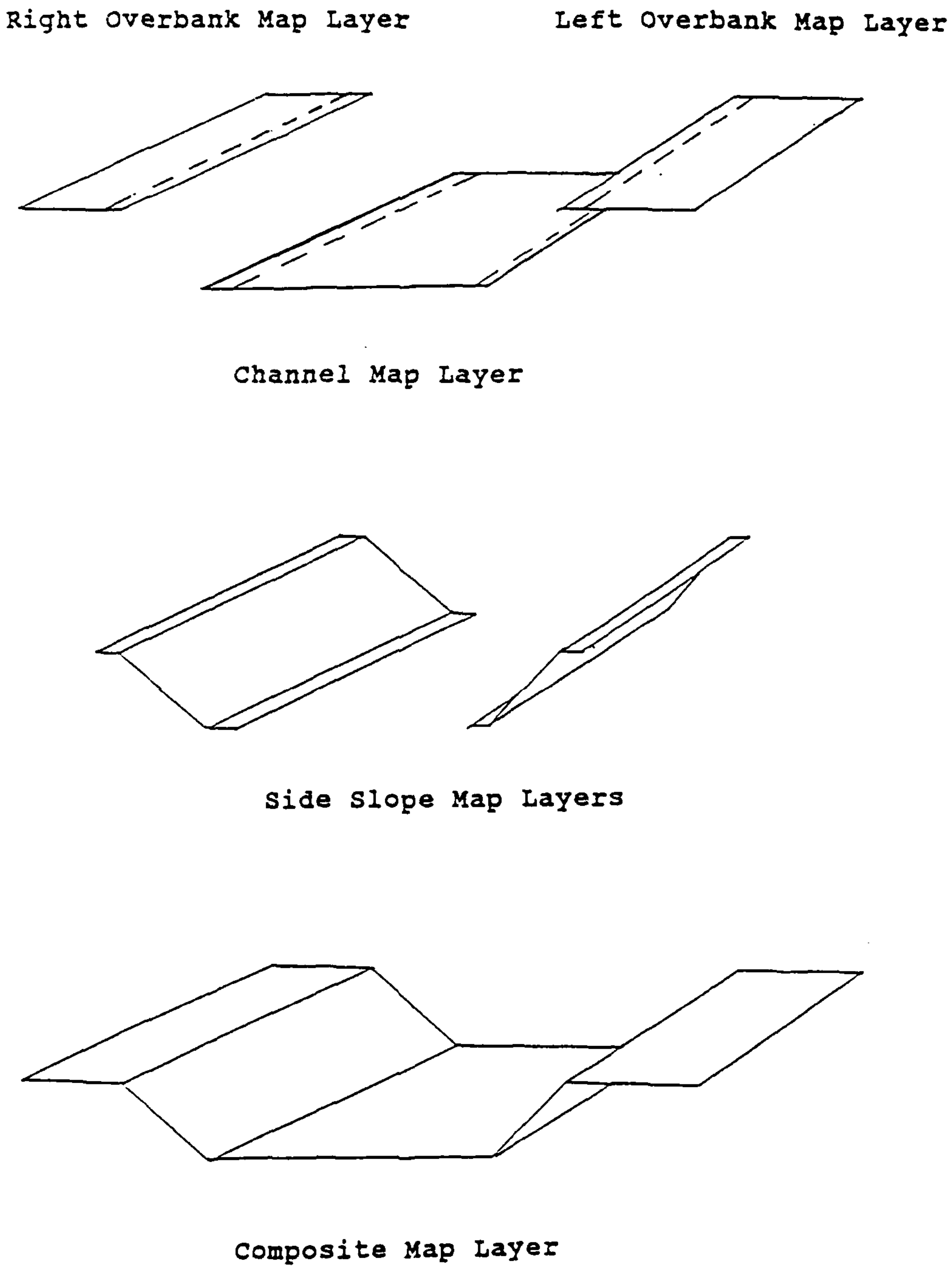


Figure 5.2 DEM construction

<u>Vehicle Selection</u>

Any of a number of amphibious vehicles, both U.S. and foreign, could have been selected for the case study. The bulk of the vehicle data base information needed to run RC-SET was available for each type. However, the composite drag coefficient was not. This is an empirical value and is dependent on the surface roughness and shape of the vehicle, it's dimensions, and orientation to the water current. One vehicle type, the U.S. Army's M2 Bradley Fighting Vehicle System (BFVS), has been recently tested for acceleration characteristics in water, and was therefore, selected for this case study. Detailed data about the vehicle was collected during ingress, egress, swim turning, and acceleration swim tests conducted by the U.S. Corps of Engineers Waterways Experiment Station (WES) near Vicksburg, Mississippi (Jones and Willoughby, 1987).

The M2 BFVS is a full tracked, lightly armored fighting vehicle designed to move mechanized soldiers around the battlefield. The overall maneuverability of the vehicle is comparable to the new main battle tank called the M1 Abrams. The BFVS has a crew of three and carries six infantrymen. Its armament includes a 25 mm automatic stabilized cannon, an anti-armor guided missile system, and a coaxial 7.62 mm machine gun. In

addition the BFVS has six firing ports located along the sides and rear of the vehicle. The M2 BFVS is shown in Figure 5.3.

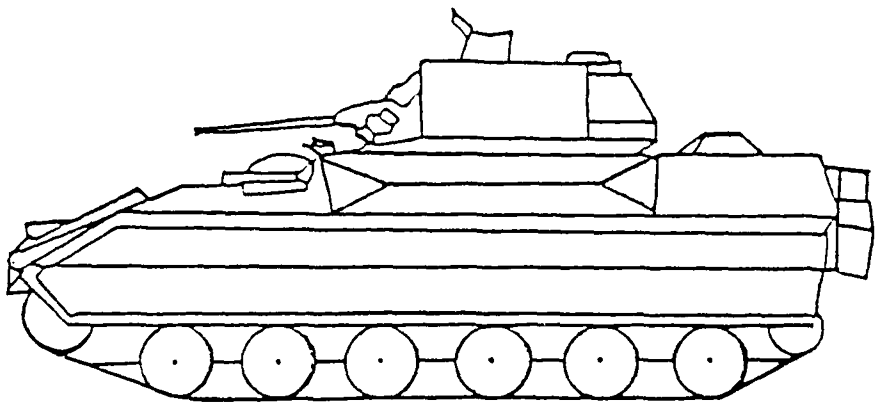


Figure 5.3 M2A1 Bradley Fighting Vehicle System (BFVS)

Due to the weight of the vehicle (approximately 60,000 lbs), it is nearly submerged when floating in water. As a result, a rubberized cloth material, called a swim curtain, is attached to the vehicle body to form a gun wall. Unfortunately, none of the vehicle's weapons can be fired with the swim curtain attached. Installation of the swim curtain by the crew takes about ten minutes. Figure 5.4 is a picture of the BFVS with the swim curtain installed.

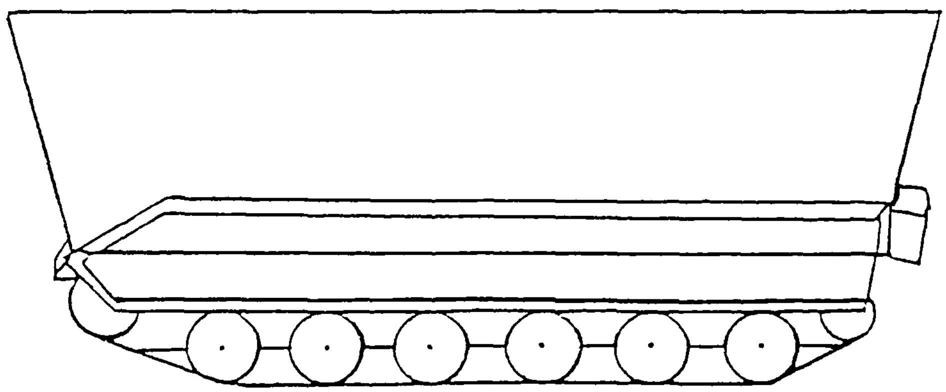


Figure 5.4 BFVS with swim curtain installed

Calibration and Validation

The digital elevation model. DEM Validation of the DEM and calibration of RMA-2D are directly related to the small amount of data available to construct the DEM. Because only ninety-six data points were used to construct a model of a study area that is 466 meters long and 336 meters wide, the DEM is a rough approximation of the real system. The model looks reasonable, and this is a first step in validation. Using a GRASS feature that

allows the user to evaluate map sheet classes, the elevation at the cross sections were checked and are correct. The interpolated cells between cross sections appear to have only minimal distortion.

The DEM map sheet is made up of a number of cells. Changes in elevation on a map sheet occur in steps and not smoothly. This is because every point in the same cell has the same elevation. This has little effect on the general relief of the channel bottom and overbanks where the difference in elevation from cell to cell is small. Such is not the case on the side slopes. The elevation gradient is great from cell to cell when moving from the bottom elevation of ten feet to the overbank elevation of fifty feet, causing a pronounced stepping in the bank slope that effects the

realism of the DEM. In summary, though it cannot be compared to detailed data of the real system, the DEM from GRASS appears to be a reasonably correct and useful.

RMA-2D. MESH was used to generate a finite element network of the reach. Figure 5.5 shows the simple network of eighty-four nodes and sixty-five elements that define the reach.

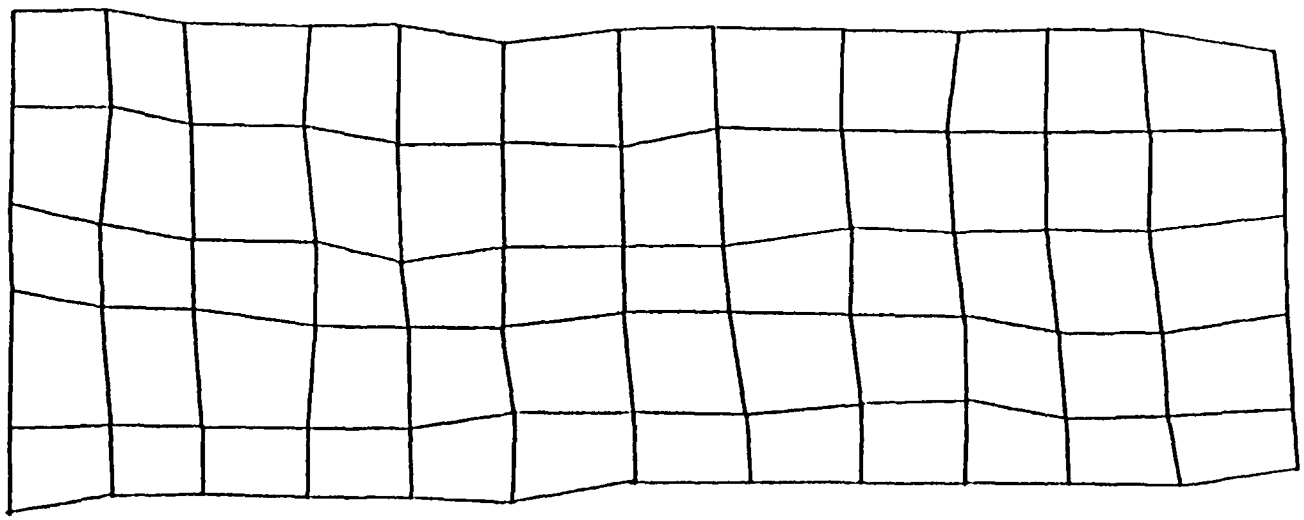


Figure 5.5 Study area finite element network

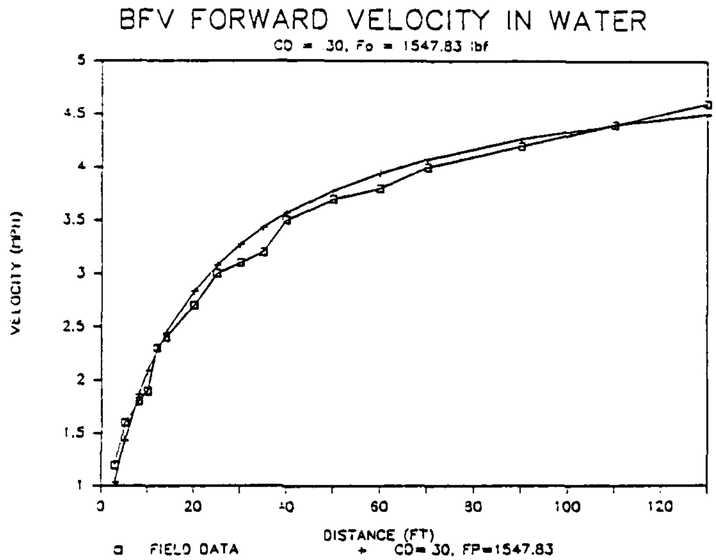
The hydrologic boundary conditions were obtained from the University of California-Davis study, though roughness and eddy viscosity coefficients were not available and had to be estimated. Velocities calculated by RMA-2D were generally five percent lower than field data measurements. This five percent was the best case, after surface roughness and eddy viscosity coefficients were adjusted. Since the finite element network is very simple, this difference in velocity can probably be contributed to errors in the DEM produced by GRASS. Five percent error is acceptable in the context of this thesis which is, after all, a feasibility study.

Amphibious vehicle movement. Ground movement validation is beyond the scope of this thesis, though the BFVS appears to move over ground in a reasonable manner during the simulation. Amphibious movement of the BFVS was partially calibrated, using curve fitting of the data available. The equation of the forces acting on the vehicle while moving through water was used to fit the acceleration data curve from the WES tests. The purpose of the curve fitting was to determine the maximum propulsive force delivered by the vehicle and a composite drag coefficient associated with the resistive forces caused by the water.

In the WES acceleration experiments on a BFVS, the vehicle started from rest in a still lake and traveled forward at full throttle until the vehicle velocity became nearly constant as the acceleration approached zero. At full throttle, the maximum propulsive force of the vehicle was applied constantly throughout the test. When a body moves through water the composite resistive forces due to the water increase in proportion to the vehicle velocity squared (White, 1986). The WES tests were conducted in both forward and reverse gears.

Equation (7), the equation determining the summation of forces acting of the vehicle, was used to evaluate the propulsive force and the drag coefficient characteristics. The two unknowns in this equation are the maximum propulsive force of the vehicle and the combined drag coefficient. Using a spread sheet computer program it was possible, through trial and error, to adjust both unknowns (the maximum propulsive force and drag coefficient) until a close fit of the force equation to the field data was achieved. A reasonable drag coefficient, based on submerged rectangles in two dimensional flow, was set as the initial value of the drag coefficient. The sum of the least squares method was used to approximate the maximum propulsive force. The magnitude of this value

was increased until the best fit of the curves occurred. The orientation of the curve was adjusted by fixing the value of the maximum propulsive force, then adjusting the value of the combined drag coefficient. Figure 5.6 shows the results of this curve fitting.



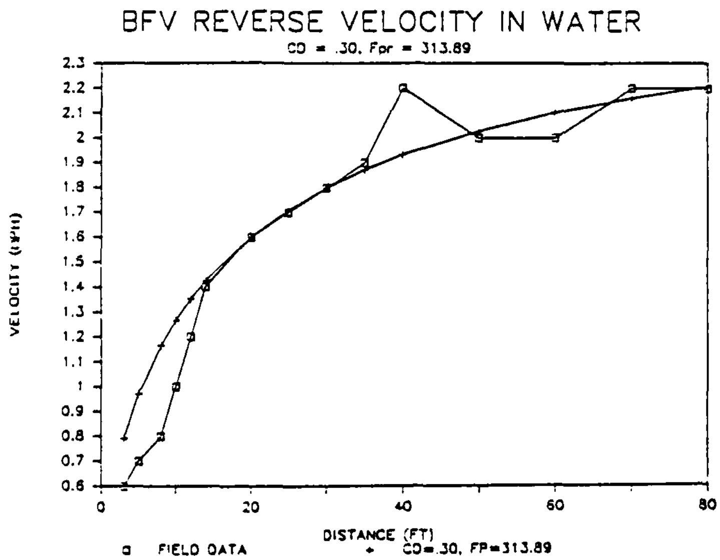


Figure 5.6 Acceleration test curve fitting

The results of this curve fitting show that the real system and the movement model are governed by the same physical laws. The model is calibrated only for amphibious vehicle movement in still water, with no turning, for both forward and reverse acceleration at full throttle. With the limited data available it is not possible to calibrate amphibious vehicle movement in moving water.

Therefore, validation of the amphibious movement simulation is limited to observation. For example, amphibious vehicles moving through water execute turns slowly because of the magnitude of resistive forces from the water relative to the propulsive power of the vehicle (U.S. Army, 1988). Amphibious vehicle movement in RC-SET also exhibits this behavior. Another observation is that the maximum velocity of the vehicle should not exceed the maximum swim speed of the vehicle plus water velocity. Such is the case in RC-SET.

Simulation Experimentation

As an example of how RC-SET can be used to evaluate river crossing techniques three crossing methods were used between the same starting point and finish line. Figure 5.7 shows the three methods (U.S. Army, 1988).

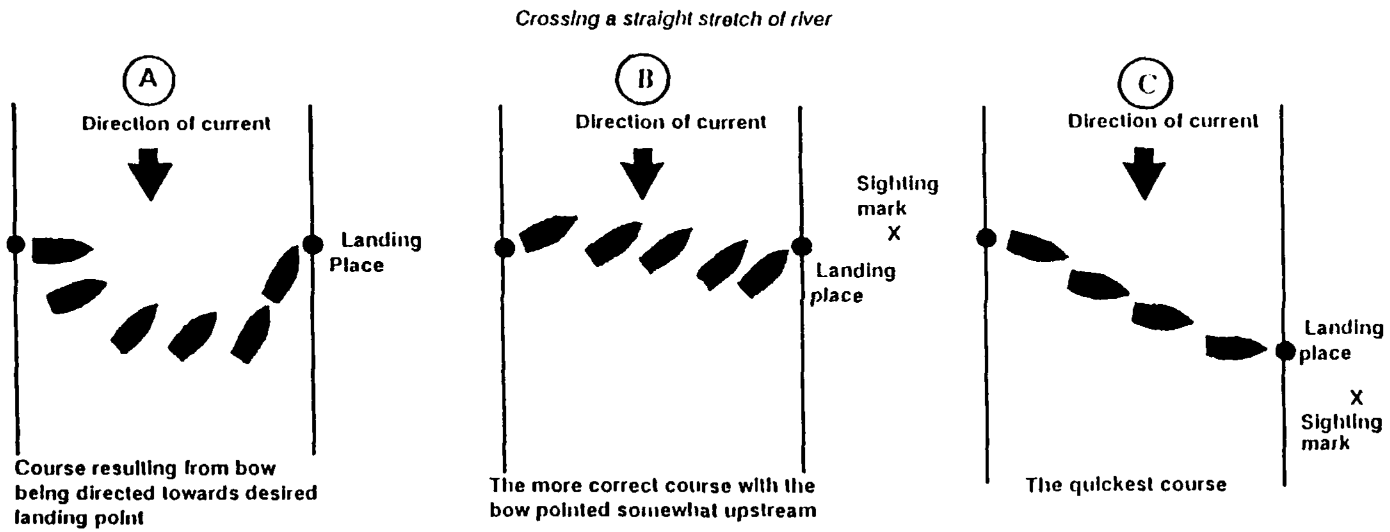


Figure 5.7 The three methods used to cross a straight reach of river

(Source: U.S. Army, 1988, Training Circular 5-210 Military Float Bridging Equipment, p. 6)

In method A the bow (front) of the vehicle is steered so that it remains pointed at the exit point on the far bank at all times. This is the slowest method because the vehicle is swept downstream by the current and will have to swim directly into the current during the last part of the crossing. Method B is faster than method A and is used when the ingress and egress points are limited to specific areas. In this method the bow of the vehicle points somewhat upstream and a nearly linear course is maintained. Controlling the vehicle is more difficult in this method because of the side slipping movement of the vehicle. Method C is the fastest of the three and is used when time in the water must be minimized and adequate egress space exists on the far bank.

The crossing time for method A was 4.38 minutes, method B 1.55 minutes, and method C 1.42 minutes. As expected, method C was the fastest.

Conclusions of the Case Study

Even though RC-SET could not be completely calibrated or validated, the case study was valuable because it confirms the thesis that a decision support system can be built to apply two dimensional hydrodynamic models to evaluate amphibious river crossing sites.

Calibration and validation of RC-SET. Although only partial calibration and validation of RC-SET was possible, the data preparation tools and components of RC-SET together produced an acceptable representation of the ersatz real system. Complete calibration is possible only through collection and application of real data from the same combination of vehicle and reach.

Usefulness of RC-SET. Some very important conclusions can be drawn from the application of RC-SET to the reach selected. RC-SET was flexible in allowing the user to build the simulation model. This allowed the user to conduct computer simulation experiments repeatedly, and vary the starting conditions each time if so desired. The user-system interface gave the user an understanding of how amphibious vehicles move through water, and how current velocity can effect maneuvering. RC-SET was also used to compare the three crossing methods. The benefit of this simulation was not only to determine the fastest crossing method, but more importantly, the user discovered how easy or difficult each method was compared to the others, in terms of controlling the vehicle against the current.

The user-system interface is easy to understand and interactive. The color shaded topography display helps the user to understand the topography. When the image of the velocity vectors is over laid on the topography, the user understands how the topography effects water velocity. This is probably the most useful feature of RC-SET. As the user drives the vehicle they learn how the two dimensional forces of the water effect the amphibious vehicle's movement.

Usefulness to the U.S. Army. RC-SET can be a valuable tool to the Corps of Engineers and the U.S. Army. As an evaluation tool, RC-SET could be used to evaluate potential river crossing sites on rivers located in friendly territory. RC-SET can also be used as a simulator to train soldiers and their leaders.

The primary application of RC-SET is to evaluate known potential crossing sites to determine if they can be crossed by amphibious vehicles. As discussed in the introduction, rivers present considerable obstacles to maneuvering forces. RC-SET can be used by friendly forces to develop a data base of amphibious vehicle crossing capabilities at specific crossing sites, at specified flow rates. This data base can be a useful planning tool for engineer

officers evaluating crossing sites that support the maneuver commander's ground tactical plan. Certainly, this will help minimize some of the risk of conducting the amphibious phase of river crossing operations.

RC-SET is not only valuable in determining the feasibility of friendly forces crossing a river. It can be used for defensive purposes as well. RC-SET can be used to determine the feasibility of enemy vehicles crossing a river. RC-SET is particularly useful along rivers that are the primary natural obstacle to impede an enemy advance. RC-SET can be a decision support tool to assist the user in selecting potential obstacle locations along the friendly shore where the enemy will most likely have to exit the water due to the topographic and hydrodynamic characteristics of the reach. Obstacle location is related to the flow rate of the water, vehicle type, and starting point selected. In this respect, RC-SET offers the friendly forces the capability of simulating enemy vehicles attempting a crossing at any point along the river where topographic and hydrologic data has been collected.

RC-SET can also be used to train soldiers and their leaders. Amphibious operations are not routinely practiced by field units because of the associated risks and increased vehicle maintenance costs.

Soldiers stationed near tactically important rivers can periodically use RC-SET to simulate swimming their vehicles without any risk. RC-SET can also be used to simulate crossings where steering is more difficult, such as crossing a river at a bend where the velocity gradient is great, as in Figure 6.1.

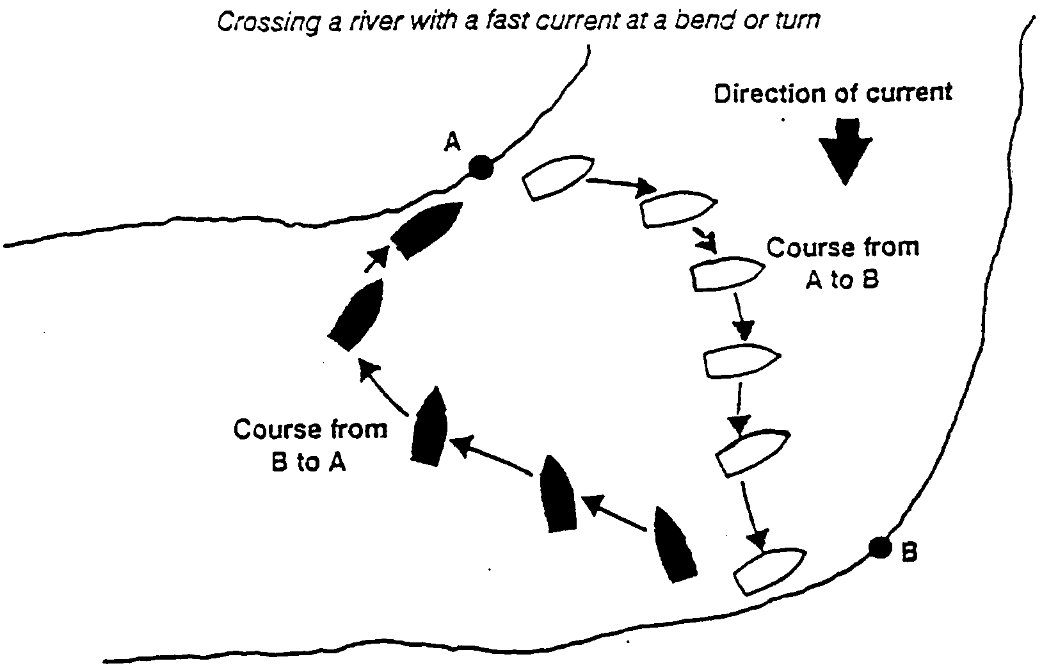


Figure 6.1 Crossing a river with a fast current at a bend or turn

(Source: U.S. Army, 1988, Training Circular 5-210 Military Float Bridging Equipment, p. 7)

Not only do soldiers become familiar with the river in their area, the gain an understanding of the behavior of amphibious vehicles maneuvering in water.

Continuing Research

RC-SET is a prototype, and not a panacea. It's success opens the door to many possible extensions of the concept of applying two dimensional hydrodynamic modeling to river crossing simulations. Continuing research could extend RC-SET type decision support systems into all phases of river crossing operations.

Amphibious operations. The focus of continuing research in this phase is to correctly model the geotechnical characteristics of the banks then integrate them into the simulation. The key characteristics are ingress and egress bank slopes and soil. Even simple GO and NO GO evaluations of bank conditions would enhance the decision support capabilities of RC-SET.

Rafting operations. RC-SET could be enhanced to include simulation of rafting operations. During rafting operations, bridge erection boats, analogous to small, shallow draft tug boats, maneuver rafts of military float bridge equipment to transport armored vehicle and anti-armor systems across a reach of river.

CHAPTER VI

RESULTS AND CONCLUSIONS

Results

RC-SET is confirmation that a decision support system can be created to apply two dimensional hydrodynamic modeling to simulate amphibious vehicle movement in water. This decision support system is then used to evaluate potential amphibious river crossing sites. RC-SET applies the results from existing DEM and hydrodynamic models in an interactive, graphical simulation. As a decision support system, it applies data from these models in a flexible, easy to understand format. In this respect, RC-SET is a useful tool to both engineers and those with limited understanding of hydrodynamics.

The format of RC-SET is structured by the three components of a decision support system. The data base management system is simple, yet capable of efficiently processing the data from GRASS and RMA-2D. The model management system is flexible and allows the user to build the simulation by selecting the crossing site, vehicle type, and river flow rate.

One, two, or three bridge erection boats are used depending on the size of the raft. Rafting can be difficult on rivers that have fast currents, rapidly changing velocities, and restrictions to loading and unloading sites. Geotechnical information will be needed to select loading and unloading sites.

Military float bridging. Perhaps the greatest enhancement of RC-SET would be the capability to evaluate bridging sites, and simulate the construction of military floating bridges. RC-SET could be used to evaluate bride erection sites along the friendly shore, simulation of bridge component erection, the rafting of bridge components to the bridge site using bridge erection boats, and finally, the construction and anchoring of the bridge. With these enhancements, RC-SET would be even more valuable as a decision support system and training tool.

Conclusion

The U.S. Army's new doctrine requires that our forces maintain the initiative on the battlefield. To do this they must have the ability to maneuver and cross or bypass any obstacles they encounter. Engineer officers supporting these forces are often hard pressed to make sound engineering judgements based upon limited information. Although uncertainty is a part of every

engineering judgement, it can be disastrous when evaluating potential river crossing sites. Sending combat engineers forward to conduct river reconnaissance, or procuring historical records helps to fill the information vacuum. Two dimensional hydrodynamic models give us the technology to effectively evaluate the magnitude and direction of current velocities through a given reach of river. If this information were available to engineer officers in the field, it would help them to locate the best river crossing site locations. This would lower the risk of selecting amphibious crossing sites with hydrologic characteristics that prohibit amphibious movement.

RC-SET is a computer simulation based decision support system that uses the results of RMA-2D to evaluate potential amphibious river crossing sites. It could provide the Corps of Engineers with a valuable tool that could be used to evaluate the hydrologic characteristics of potential amphibious crossing sites before the conflict begins. The results of RC-SET could be published and distributed to field engineer units. During the conflict the published data base could be used to help identify the best crossing locations for friendly forces, or in the defense, identify the most likely locations of enemy river crossing operations. Additionally, RC-SET opens the

door for future enhancements and uses that would make it a valuable training and evaluation tool.

RC-SET is a prototype. It has only been calibrated and validated for specific functions, such as forward and reverse acceleration in still water. Simulations using RC-SET have not been compared to data collected from field experiments of actual vehicles attempting to cross a specific reaches. However, partial calibration and validation of the prototype confirms the thesis that a decision support system can be developed that uses the results of a two dimensional hydrodynamic model to evaluate potential amphibious crossing sites.

REFERENCES

Andriole, Stephen J., (ed.) (1986) Microcomputer Decision Support Systems: Design, Implementation, and Evaluation; Wellesly: QED Information Systems.

Bedient, Philip B., and Huber, Wayne C. (1988) Hydrology and Flood Plain Analysis; Reading: Addison-Wesley Publishing Company.

Cellier, Francois E. (1982) Progress in Modelling and Simulation; New York: Academic Press.

Chapra, Steven C., and Canale, Raymond P. (1986) Introduction to Computing for Engineers; New York: McGraw-Hill Book Company.

Chapra, Steven C., and Canale, Raymond P. (1988) Numerical Methods for Engineers; New York: McGraw-Hill Book Company.

Chow, Ven Te. (1988) Open-Channel Hydraulics; New York: McGraw-Hill Book Company.

Chow, Ven Te, Maidment, David R., and Mays, Larry W. (1988) Applied Hydrology; New York: McGraw-Hill Book Company.

Daugherty, Robert L., and Franzini, Joseph B. (1977) Fluid Mechanics With Engineering Applications; New York: McGraw-Hill Book Company.

Filstrup, A. W. III, (ed.) (1988) Computer-Aided Engineering Applications; New York: American Society of Mechanical Engineers.

Froehlich, David C. (1988) Finite Element Surface-Water Modeling System: Two-Dimensional Flow in a Horizontal Plane, Volume 1 - User's Manual; Reston: U.S. Geological Survey.

Hopple, Gerald W. (1988) The State of the Art in Decision Support Systems; Wellesly: QED Information Systems.

Gee, D. Michael (1986) Use of a Two-Dimensional Flow Model to Quantify Aquatic Habitat; Paper presented to the American Society of Civil Engineers Specialty Conference on Computer Applications in Water Resources; Buffalo, New York; 10-12 June 1985.

Gee, D. Michael (1985) Role of Small Computers in Two-Dimensional Flow Modeling; Paper presented at the American Society of Civil Engineers Hydrualics Division Specialty Conference on Hydraulics and Hydrology in the Small Computer Age; Orlando, Florida; 12-17 August 1985.

Gillmer, Thomas C. (1970) Modern Ship Design. Annapolis: United States Naval Institute.

Jones, Randolph A., and Willoughby, William E. (1987) Ingress/Egress and Swim Tests with the High Survivability Variant of the M2 Bradley Fighting Vehicle System; Vicksburg: USA-WES.

King, Ian P. (1988) A Users Guide for the Combined Element Version of RMA-2V, Release 4.1.; Lafayette: Resource Management Associates.

King, Ian P., and Roig, L. C. (1988) Two-Dimensional Finite Element Models for Flood Plains and Tidal Flats; Paper presented to the International Conference on Computational Method in Flow Analysis; Okayama, Japan; 5-8 September 1988.

Koffman, Elliot B., and Friedmann, Frank L. (1987) Problem Solving and Structured Programming in FORTRAN 77; Reading: The Addison-Wesley Publishing Company.

Lewis, T. G., and Smith, B.J. (1979) Computer Principles of Modeling and Simulation; Boston: Houghton Mifflin Company.

Meriam, J. L. (1978) Dynamics; New York: John Wiley & Sons.

Negoita, Constantin V. (1987) Simulation, Knowledge-Based Computing, and Fuzzy Statistics; New York: Van Nostrand Reinhold Company.

Over, Thomas, and Zagona, Edith (1989) An Interactive Graphical Mesh Design System For 2-D Finite Element Analysis of Surface Water Flow; Report presented as a term project in CE 6837 Computer Graphics and Computer Aided Design; University of Colorado, Boulder, Colorado; May 1989.

Pidd, Michael (1988) Computer Simulation in Management Science; New York: John Wiley and Sons.

Poppv, E. P. (1976) Mechanics of Materials; Englewood: Prentice-Hall.

Sears, Francis W., Young, Hugh D., and Zemansky, Mark W. (1976) University Physics; Reading: Addison-Wesley Publishing Company.

Tietjens, O. G. (1957) Applied Hydro- and Aeromechanics; New York: Dover Publications.

Timoshenko, S., and Young, D. H. (1968) Elements of Strength of Materials; New York: D. Van Nostrand Company.

Tipnis, V. A., (ed.), and Patton, E. M., (ed.) (1988) Computers in Engineering 1988, Volume 2; New York: The American Society of Mechanical Engineers.

Tipnis, V. A., (ed.), and Patton, E. M., (ed.) (1988) Computers in Engineering 1988, Volume 3; New York: The American Society of Mechanical Engineers.

Tod, I. C., and DeVries, J. J. (1987) Physical Model Investigations of River Control Structures, Final Report; University of California, Davis, Water Science and Engineering Paper No. 1083.

Tupper, E. C., Rawson, K. J. (1968) Basic Ship Theory; New York: American Elsevier Publishing Company.

Turabian, Kate L. (1973) A Manual For Writers of Term Papers, Theses, and Dissertations; Chicago: The University of Chicago Press.

U.S. Congress, Senate, Armed Services Committee (1989) The Posture of the United States Army, Fiscal Years 1990/1991; 101st Congress, 1st session.

U.S. Department of the Army, Corps of Engineers (1987) Users and Programmers Manual For the Geographical Resources Analysis Support System; USA-CERL, ADP Report N-87/22.

U.S. Department of the Army (1988) Training Circular Number 5-210, Military Float Bridging Equipment; Washington, D.C.: Government Printing Office.

U.S. Department of the Army (1984) Field Manual Number 5-100, Engineer Combat Operations; Washington, D.C.: Government Printing Office.

U.S. Department of the Army (1985) Field Manual Number 5-36, Route Reconnaissance and Classification; Washington, D.C.: Government Printing Office.

U.S. Department of the Army (1978) Field Manual Number 21-33, Terrain Analysis; Washington, D.C.: Government Printing Office.

White, Frank M. (1986) Fluid Mechanics; New York: The McGraw-Hill Book Company.